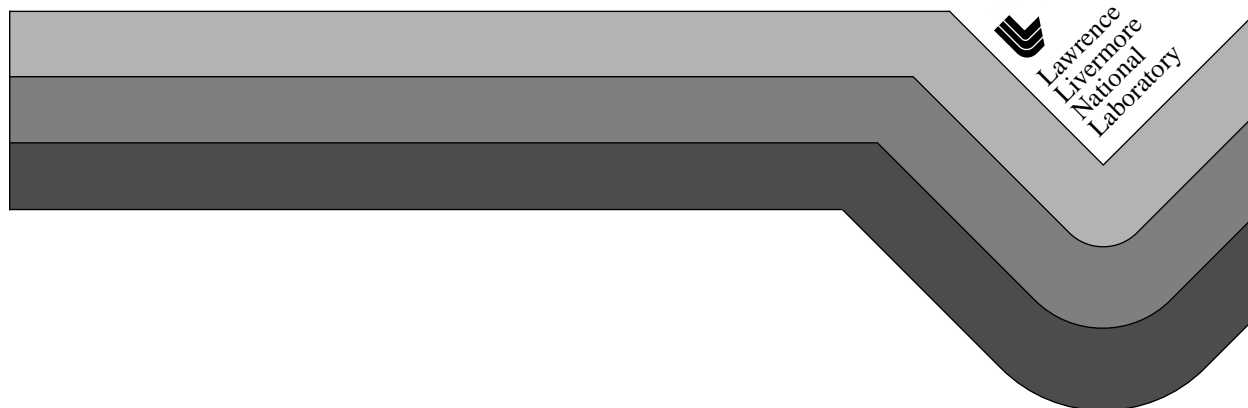


MeshTV User's Manual

March 2002

Version 4.3



DISCLAIMER

This document was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor the University of California nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or the University of California. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or the University of California, and shall not be used for advertising or product endorsement purposes.

Work performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract W-7405-ENG-48.

Introduction to MeshTV	1
Overview	1
MeshTV windows	2
Manual conventions	2
Manual chapters	2
On-line help	3
Glossary	3
Summary	3
Main Window	5
Overview	5
Selected files area	7
Overview	7
Selected files list	7
Open button	7
Replace button	8
Overlay button	8
Output window icon	8
Output window	9
Busy clock icon	9
Animation area	9
Overview	9
Animation slider	10
Animation slider text field	10
Reverse button	10
Back button	10
Stop button	10
Play button	10
Advance button	11
Active plots area	11
Overview	11
Active window	11
Maintain limits label and toggles	12
Replace plots toggle button	12
Auto update toggle button	12
Active plots list	12
Delete button	12
Draw button	13
Hide/Show button	13
Apply operators to all plots toggle button	13
Notepad area	13
Overview	13
Plots, Plot Attributes, and Variables	15
Overview	15
Plots menu	16
Overview	16
Block plot	16

- Boundary plot 17
- Contour plot 17
- Filled boundary plot 18
- Label plot 18
- Mesh plot 19
- Pseudocolor plot 19
- Surface plot 20
- Vector plot 20
- Plot attributes menu 21
 - Overview 21
- Similar items 21
 - Apply button 22
 - Dismiss button 22
 - Fill type radio buttons 22
 - Legend toggle button 22
 - Line color button 23
 - Line style menu 23
 - Line width menu 23
 - Log button 23
 - Make default button 23
 - Multiple button 23
 - Post button 23
 - Reset button 24
 - Single button 24
- Block plot attributes 24
 - Block plot attributes menu item 24
 - Block plot attributes window 25
 - Point size text field 25
 - Separate toggle button 25
- Boundary plot attributes 25
 - Boundary plot attributes menu item 25
 - Boundary plot attributes window 26
 - Boundary color buttons 26
- Contour plot attributes 26
 - Contour plot attributes menu item 26
 - Contour plot attributes window 27
 - Contour color buttons 27
 - Select by options 27
 - Limits toggle button 28
 - Min text field 28
 - Max text field 28
- Curve plot attributes 28
 - Curve plot attributes menu item 28
 - Curve plot attributes window 29
 - Labels toggle 29
- Filled boundary plot attributes 29

- Filled boundary plot attributes menu item 29
- Filled Boundary plot attributes window 30
- Material color buttons 30
- Zone type radio buttons 30
- Label plot attributes 30
 - Label plot attributes menu item 30
 - Label plot attributes window 31
 - Nodes toggle button 31
 - Node offset label and text field 31
 - Zones toggle button 32
 - Zone offset label and text field 32
 - Flip button 32
 - String label and text field 32
 - Number of labels to show toggle and text field 32
 - Horizontal justification option menu 33
 - Vertical justification option menu 33
 - Text height text field 33
- Mesh plot attributes 33
 - Mesh plot attributes menu item 33
 - Mesh plot attributes window 34
 - Outline only toggle 34
 - Error text field 34
 - Point size 34
- Pick plot attributes 35
 - Pick plot attributes menu item 35
 - Pick plot attributes window 35
 - Text color button 35
- Pseudocolor plot attributes 35
 - Pseudocolor plot attributes menu item 35
 - Pseudocolor plot attributes window 36
 - Centering toggles 36
 - Limits toggle 36
 - Min text field 37
 - Max text field 37
 - Scale radio buttons 37
 - Skew factor text field 37
 - Lighting toggle 37
 - Point size text field 38
- Reference line plot attributes 38
 - Reference line plot attributes menu item 38
 - Reference line plot attributes window 39
 - Endpoints area 39
 - Absolute endpoints radio button 39
 - Logical endpoints radio button 39
- Surface plot attributes 40
 - Surface plot attributes menu item 40

- Surface plot attributes window 40
- Surf color button 41
- Wire color button 41
- Limits toggle 41
- Min text field 41
- Max text field 41
- Surface toggle 42
- Wire frame toggle 42
- Vector plot attributes 42
 - Vector plot attributes menu item 42
 - Vector plot attributes window 42
 - Scale text field 43
 - Head size text field 43
 - Nvectors text field 43
- Variables menu 43
 - Overview 43
- Operators and Operator Attributes 45
 - Overview 45
 - Operators menu 46
 - Overview 46
 - Arbitrary slice operator 46
 - Erase operator 46
 - Index select operator 46
 - Material selection operator 46
 - Orthogonal slice operator 47
 - Reflect plot operator 47
 - Segment operator 47
 - Remove last operator 47
 - Remove all operators 47
 - Operator attributes menu 47
 - Overview 47
- Similar items 48
 - Apply button 48
 - Dismiss button 48
 - Make default button 49
 - Post button 49
 - Reset button 49
- Arbitrary slice operator attributes 49
 - Arbitrary slice operator attributes menu item 49
 - Arbitrary slice attributes window 50
 - Origin text field 50
 - Normal text field 50
 - Up axis text field 50
 - Map to 2D toggle button 51
 - Copy slice plane button 51
- Erase operator attributes 51

- Erase operator attributes menu item 51
 - Erase attributes window 52
 - Plane option menu 52
 - Status radio buttons 52
 - Longitude slider bar and text field 52
 - Latitude slider bar and text field 53
 - Radius slider bar and text field 53
 - Offset text field 53
- Index select operator attributes 54
 - Index select operator attributes menu item 54
 - Index select attributes window 54
 - Dimension radio buttons 54
 - Min, max, incr text fields 55
 - Hierarchy radio buttons 55
- Material selection operator attributes 55
 - Material selection operator attributes menu item 55
 - Material selection attributes window 56
 - Active materials list 56
 - All button 56
 - None button 56
- Orthogonal slice operator attributes 57
 - Orthogonal slice operator attributes menu item 57
 - Orthogonal slice attributes window 57
 - Axis radio buttons 57
 - Slice by label and radio buttons 58
 - Slice by percent 58
 - Slice by coordinate 58
 - Slice by zone 58
- Reflect plot operator attributes 59
 - Reflect plot operator attributes menu item 59
 - Reflect plot attributes window 59
 - Positive Z 59
 - Negative Z 60
- Segment operator attributes 60
 - Segment operator attributes menu item 60
 - Segment attributes window 61
 - Initial zone area 61
 - Zone text field 61
 - Block text field 61
 - Use picked zone toggle 61
 - Layers slider and text field 62
- File Menu 63
 - Overview 63
 - Select file 65
 - Overview 65
 - Select file window 65

- Path text field 65
- Filter text field 66
- Directories list 66
- Files list 66
- Select button 66
- Select all button 67
- Selected files list 67
- Remove button 67
- Remove all button 67
- Check for directories within files button 67
- OK button 67
- Cancel button 68
- Print window 68
- Set print options 68
 - Overview 68
 - Set print options window 68
 - Printer command radio buttons 69
 - Printer text field 69
 - Printer type 69
 - Raster postscript radio button 69
 - Postscript radio button 69
 - Print tiled toggle button 70
 - Screen capture toggle button 70
 - X/Y resolution text field 70
 - Banner text field 70
 - Print button 70
 - Apply button 71
 - Dismiss button 71
- Save window 71
- Set save options 71
 - Overview 71
 - Set save options window 72
 - Name text field 72
 - File type 72
 - Raster postscript radio button 72
 - Postscript radio button 73
 - Ppm radio button 73
 - RGB radio button 73
 - Tiff radio button 73
 - STL radio button 73
 - Family toggle button 74
 - Autosave toggle button 74
 - Save tiled toggle button 74
 - Screen capture toggle button 74
 - Resolution 74
 - Maintain aspect toggle button 75

- Banner text field 75
 - Save button 75
 - Apply button 75
 - Dismiss button 75
- Save movie 75
 - Overview 75
 - Save movie window 76
 - File name text field 76
 - File type option 76
 - Generate movie now toggle button 76
 - Frame options 76
 - Start frame text field 77
 - End frame text field 77
 - Increment by text field 77
 - Resolution 77
 - Generate button 77
 - Movie tips button 77
 - Dismiss button 77
- Movie tips 78
 - Movie tips window 78
- Save settings 78
- Simulation 79
 - Overview 79
 - Simulation window 79
 - Simulations list 79
 - Pause button 80
 - Continue button 80
 - Dismiss button 80
- Quit 80
- Controls Menu 81
 - Overview 81
 - Similar items 83
 - Apply button 83
 - Dismiss button 84
 - Post button 84
 - Animation 84
 - Overview 84
 - Animation window 84
 - Cache toggle button 84
 - Speed Toggle button 85
 - Speed text field 85
 - Speed slider bar 85
 - Annotation 85
 - Overview 85
 - Annotation window 86
 - General options 86

- User info toggle button 86
 - Database info toggle button 87
 - Plot legends toggle button 87
 - Viewport type options 87
 - Viewport text field 87
 - Verbose label and option menu 87
 - Legend background color buttons 87
- Banner options 88
 - Banner display toggle button 88
 - Banner text field 88
 - Banner location text field 88
 - Banner height text field 88
 - Banner color button 88
- 2D axes 88
 - 2D axis labels options 89
 - 2D grid lines options 89
 - 2D tick marks options 89
 - 2D tick mark locations options 89
- 3D axes 89
 - 3D axis labels options 89
 - 3D grid lines options 89
 - 3D tick marks options 90
 - 3D tick mark locations options 90
 - 3D triad radio buttons 90
- Viewgraph button 90
- Reset Button 90
- Block 90
 - Overview 90
 - Block window 91
 - Turn all on button 91
 - Turn all off button 91
 - Turn on textfield 91
 - On button 91
 - Turn off textfield 92
 - Off button 92
 - Active blocks list 92
 - Problem block range label 92
- Color table 92
 - Overview 92
 - Color table window 93
 - Color table manager 93
 - Color table selection option 93
 - Cale black option 93
 - Cale white option 94
 - Contoured option 94
 - Gray scale option 94

- Hot to cold option 94
- Rainbow option 94
- X-ray option 94
- User option 94
- File name text field 95
- New button 95
- Delete button 95
- Save to file button 95
- MeshTV default button 95
- Color table editor 96
- Color spectrum 96
- Add color button 96
- Remove color button 96
- Align button 96
- Equal spacing toggle button 97
- Smooth toggle button 97
- Color value scroll bar 97
- Color value text field 97
- Expressions 97
 - Overview 97
 - Expression window 98
 - Name text field 98
 - Expression text field 98
 - Type option 99
- Groups 99
 - Overview 99
 - Group window 100
 - Active groups list 100
 - All button 100
 - None button 100
- Lighting 100
 - Overview 100
 - Lighting window 101
 - Lighting layout 101
 - Light selection 101
 - Light properties 102
 - Light preview 102
- Line-out 103
 - Overview 103
 - Line-out window 103
 - Reference line plot attributes area 103
 - Logical toggle button 104
 - Replace on replot toggle button 104
 - Curve plot attributes area 104
 - Output window label and text field 104
 - Use 1st window toggle button 104

- Replace on replot toggle button 105
- Materials 105
 - Overview 105
- Palette editor 105
 - Overview 105
 - Palette editor window 106
 - Select mode label 106
 - Output window colors 106
 - Background color 106
 - Foreground color 107
 - Colors 107
 - Color value 107
 - Color text 107
 - Lookup color by name 108
 - MeshTV default button 108
- Powerwall 108
 - Overview 108
 - Powerwall window 109
 - Powerwall on 109
 - Powerwall replot 109
 - Layout area 109
 - Mapping 110
 - Token 110
 - Token tray 110
 - Mapping area 110
 - Unmap all button 110
- Species 111
 - Overview 111
 - Species window 111
 - Active species list 111
 - All button 111
 - None button 112
- Extra Menu 113
 - Overview 113
 - Command line 114
 - Overview 114
 - Command line interface window 114
 - Command line scrolled window 114
 - Command text field 115
 - Post/Unpost button 115
 - Dismiss button 115
 - Command log 115
 - Overview 115
 - Command log window 116
 - Dismiss 116
- Preferences 116

- Overview 116
 - Preferences window 117
 - Information Display area 117
 - Display hints toggle button 117
 - Display messages toggle button 117
 - Display warnings toggle button 117
 - Miscellaneous area 118
 - Show hidden variables toggle button 118
 - Renderer area 118
 - Optimize for 2D graphics toggle button 118
 - Depth cueing controls 118
 - Material interface reconstruction controls 118
 - Subdivision level 119
 - Maximum iterations 119
 - Use old isosurface algorithm 119
 - Volume fraction 119
 - Opening files area 119
 - Reset attributes on replace toggle button 119
 - Ignore file times toggle button 120
 - Use current working directory toggle button 120
 - Apply button 120
 - Dismiss button 120
- Window Menu 121
 - Overview 121
 - Add 122
 - Delete 122
 - Clear all 122
 - Flip Main window 122
 - Layout 123
 - Overview 123
 - 1x1 123
 - 2x1 123
 - 2x2 123
 - 2x4 124
 - 3x3 124
 - 4x4 124
- Help Menu 125
 - Overview 125
 - On help 126
 - Overview 126
 - Help window 127
 - Help on 127
 - Help text 127
 - User notes 128
 - Dismiss button 128
 - Getting Started 128

- Overview 128
- User's Manual 128
 - Overview 128
- Command Line Interface 128
 - Overview 128
- Release notes 129
 - Overview 129
 - Release notes window 129
 - Display areas 130
 - Chapter 130
 - Enhancements and Modifications 130
 - Corrected Bugs 130
 - Workarounds for Known Bugs 130
 - Section 131
 - Dismiss button 131
- Copyright 131
 - Overview 131
 - Copyright window 132
 - Dismiss button 132
- Credits 132
 - Overview 132
 - Credits window 133
 - Dismiss button 133
- Information window 134
 - Information window 134
 - Dismiss button 134
 - Read release notes button 134
- Query window 134
 - Information window 134
- Visualization Windows 135
 - Overview 135
 - Window modes 136
 - Navigate mode 136
 - Zoom mode 136
 - Lineout mode 136
 - Pick mode 137
 - Slice pick mode 137
 - Slice mode 137
 - The Popup menu 138
 - Overview 138
 - Mode menu 140
 - Clear 140
 - Reset view 140
 - Redraw 140
 - Choose center 141
 - Clear pick pts 141

- Print image 141
- Save image 141
- Copy menu 142
 - View from 142
 - Lighting from 142
 - Annotations from 142
 - Palette from 143
 - Everything from 143
- Display menu 144
 - Invert background 144
 - Navigate bbox 144
 - Display toolbar 144
 - Perspective 144
 - Spin 145
 - Stereo 145
- Flip xy toggle 145
- Full frame toggle 145
- The toolbar 146
 - Overview 146
 - Collapse/Expand button 147
 - Mode buttons 147
 - Reset view button 147
 - Lock button 147
 - Activate button 148
- Glossary 149
 - Table of definitions 149

Chapter 1

Introduction to MeshTV

1.0 Overview

MeshTV is an interactive program that visualizes and analyzes scientific data. MeshTV reads SILO¹ data files, which allows it to run on many hardware platforms. MeshTV handles 1D, 2D, and 3D data and provides a variety of data operations, enabling it to be both a visualization tool and a data analysis program. This manual covers the MeshTV Graphical User Interface (GUI). A full description for using the SILO data format with MeshTV can be found in the *SILO-MeshTV Manual*², and instructions for writing SILO files can be found in the *SILO User's Guide*³.

While MeshTV has a Command Line Interface (CLI)⁴, most people prefer to use MeshTV's GUI. If you are just starting to use MeshTV for the first time, you might want to read the *MeshTV Getting Started Manual*⁵, which contains tutorials and descriptions of important MeshTV assumptions. If you are interested in the Command Line Interface, please see the *MeshTV Command Line Interface Manual*.

This manual details the capabilities provided by the MeshTV GUI, explaining the function of various options in the different windows. It also describes the GUI elements of the visualization windows. It is broken into chapters, as described in Section 4.

-
1. A library developed at Lawrence Livermore National Laboratory to handle scientific database issues, such as cross-platform compatibility, mesh, material, and variable data structures, and multidimensional arrays.
 2. *SILO-MeshTV Manual*, To be written.
 3. *SILO User's Guide*, UCRL-MA-118751
 4. *MeshTV Command Line Interface Manual*, UCRL-MA-127443. See Chapter 1, Section 5 to learn how to access this manual from within MeshTV.
 5. *MeshTV Getting Started Manual*, UCRL-MA-127442. See Chapter 1, Section 5 to learn how to access this manual from within MeshTV.

2.0 MeshTV windows

MeshTV contains a minimum of two windows. These are the **Main** window titled “MeshTV” and a visualization window titled “Window 1.” (There can be up to 16 windows of the same type as Window 1, and each window displays a different number.)

Most subwindows are accessed from the **Main** window via menu selections which contain a ‘...’ after the text. Buttons or menus with the ‘...’, like **Select files...**, indicate that a window will pop up when the button is pressed or the menu item is selected.

3.0 Manual conventions

This manual uses the following conventions:

Element	All GUI elements, like windows, menus, and buttons, will use bold helvetica .
<i>Documents</i>	All document names will be italicized.

4.0 Manual chapters

This manual contains the following chapters:

Introduction	This chapter.
Main Window	Descriptions of the MeshTV Main window.
Plots	Descriptions of the various plots and the plot attribute windows and of the Variable pulldown menu.
Operators	Descriptions of the various operators and the operator attribute windows.
File Menu	Descriptions of the File menu and any windows or operations which are accessed via the File menu, like the Set print options window.
Controls Menu	Descriptions of the Controls menu and any windows or operations which are accessed via the Controls menu, like the Animate window.
Extra Menu	Descriptions of the Extra menu and any windows or operations which are accessed via the Extra menu, like the Preferences window.
Window Menu	Descriptions of the Window menu and any windows or operations which are accessed via the Window menu, like the Add operation.

Help Menu	Descriptions of the Help menu and any windows or operations which are accessed via the Help menu, like the Help window.
Visualization windows	Descriptions of the visualization windows and operations which are accessed via these windows, like the toolbar.
Glossary	Definitions of words and concepts found in this manual.

5.0 On-line help

Note that to use MeshTV's online help, you must have a 3 button mouse.

Most of this manual exists as on-line help, accessible directly from within MeshTV. To use this capability, run MeshTV and position your mouse cursor over the item (button, text field, label, radio button, etc.) for which you want help. Press the right mouse button. The **Help** window will pop up, displaying any help which is available for that particular item. The information displayed will match the text for the item in question from the most current version of this document.

Material for the on-line help comes directly from this document, the *MeshTV User's Manual*, which can be accessed via Netscape Navigator by selecting the **User's Manual...** item from the **Help** pulldown menu. Likewise, you can access the *MeshTV Getting Started Manual* and the *MeshTV Command Line Interface Manual* via Netscape by going to the **Help** pulldown menu and selecting **Getting Started...** and **Command Line Interface...** respectively.

6.0 Glossary

Certain terms contain specific meanings when used in a scientific visualization arena. If you are confused about a word, check for a definition in the Glossary, which occurs at the end of this manual.

7.0 Summary

This manual covers the MeshTV GUI, but it doesn't really cover MeshTV concepts or demonstrate how to use the program. If this is your first time using MeshTV, be sure to request a copy of the *MeshTV Getting Started Manual* so you can try out the tutorials and read up on some basic MeshTV concepts. Or you can access the manual via Netscape Navigator by selecting the **Getting Started...** item in the **Help** pulldown menu.

Chapter 2

Main Window

1.0 Overview

This chapter covers the items found in the MeshTV **Main** window. It does not cover the items in the **Main** menu bar at the top of the window, nor does it cover the **Plots** pulldown, **Plot attributes** pulldown, **Operators** pulldown, **Operator attributes** pulldown, or **Variables** pulldown. All the items which aren't covered here are covered in other chapters.

Most operations in MeshTV are accessed via the **Main** window. Files can be opened, plots can be drawn and animated, posted windows can be manipulated, and operations like reflect and slice can be performed on plots.

While plots and operators are accessed from this window, they are detailed enough to require their own chapters. See Chapter 3 for a discussion of plot types and their attributes, and Chapter 4 for a discussion of operators and their attributes.

At times you will need to open windows to perform other operations. Those windows and operations are described in other chapters. Operations which deal with files and saving and printing images (things under the **File** pulldown menu) can be found in Chapter 5. Chapter 6 deals with operations which are problem-wide rather than plot-specific (and which are found under the **Controls** pulldown menu). The functionality found under the **Extra** pulldown menu can be found in Chapter 7. To find help on operations involving visualization windows, or items under the **Window** pulldown menu, look in Chapter 8. To find out more about using the on-line help system, or to read the copyright, release notes, or the list of credits, check out Chapter 9.

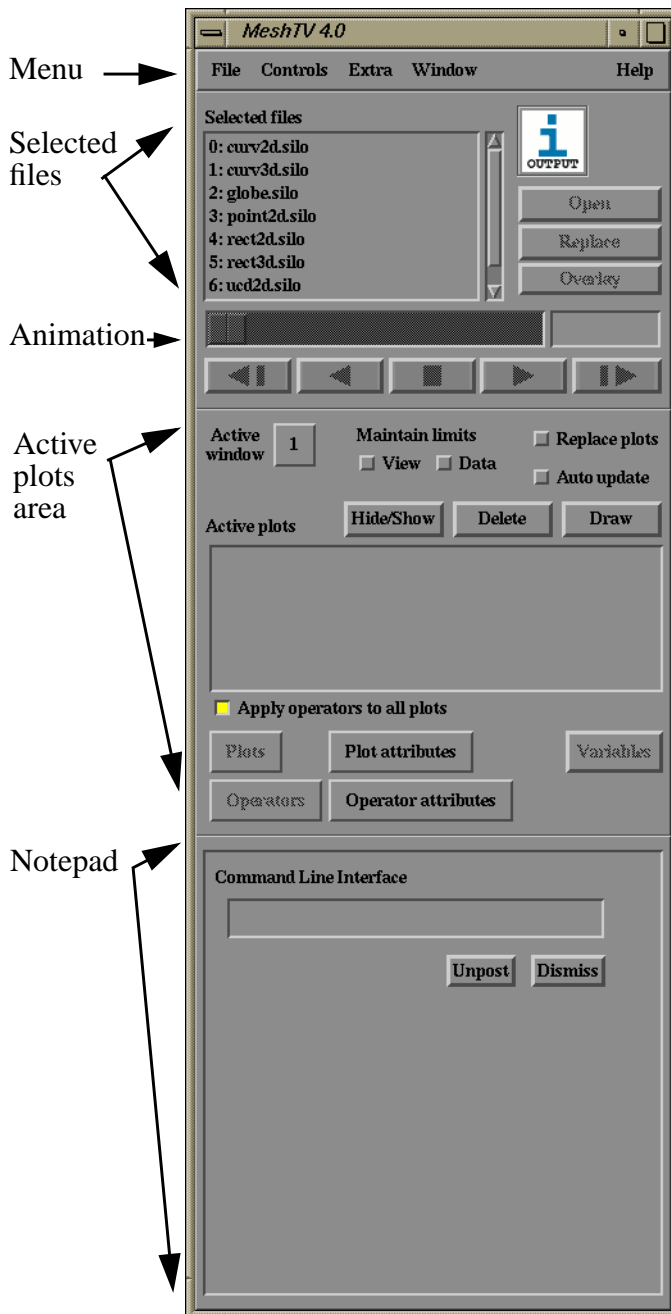


Figure 2-1: Main window

new plot replace the existing plots, and directs MeshTV to automatically update plots when their attributes change. The area lists the current plots and allows deleting and drawing of plots. It also allows you to choose which plots to draw and to specify attributes for those plots. It provides a mechanism for specifying operations on plots and for setting attributes for those operations. You can also indicate if you want to apply operators to all plots or just to the selected plots.

Notepad

Figure 2-1 shows the standard MeshTV **Main** window. The window might look slightly different when you pull it up if you have saved your own preferences. The files listed in the **Selected files** list will probably be different, too.

There are several sections to this window. These are the **Main** menu bar, the **Selected files** area, the **Animation** area, the **Active plots** area, and the **Notepad** area.

Main menu bar

This area provides pulldown menus which allow you to access many MeshTV capabilities.

Selected files

This area allows you to open files which have already been selected.

Animation

This area provides controls to animate plots from open files.

Active plots

This area allows specification of the active visualization window, provides the ability to have a

If you plan to use a window repeatedly, but you'd prefer not to have it constantly open and cluttering up your screen, you might want to post it to this area, if you can. Windows which can be posted will have a **Post** button, and once you press that button, the window will appear in this area. On lower-resolution screens, this area can be too small to fully view the posted windows.

2.0 Selected files area

2.1 Overview

This area, shown in Figure 2-2, provides a quick interface to your data files. This area consists of the **Selected files** list, the **Open** button, the **Replace** button, the **Overlay** button, and the **Output window** icon, and the **Busy clock** icon, which is only visible when MeshTV is working on displaying a plot.



Figure 2-2: Selected files area

2.2 Selected files list

This scrollable list displays selected files. When you start MeshTV, this list will usually already contain files. These are the files in the directory that matched the path stored in the **Select files** window (accessed via the **File** pulldown menu in the MeshTV **Main** window) and that also matched the file filter found in that window. If you want to select other files, you must go to the **Select files** window by pulling down the **File** menu and choosing the **Select files...** entry. When you select new files via that window, the selected files appear in this list. When you click on a file name, the **Open**, **Replace**, and **Overlay** buttons will become active. You can also open a file by double clicking on the file name.

Each file in this list receives a number before it, like “0: myfile.silo”, and this number allows you to associate variables in a plot with their parent data file. All plots in the **Active plots** list contain a number before them which corresponds to the number of the file from which the variable came. For example, “0: MESH - mesh1” indicates that the variable, mesh1, came from the “myfile.silo” file.

2.3 Open button

Use this button to open a file in the **Selected files** list. This button will be grayed out until you click on a file in the list. Once you open a file, various other options (like

creating plots) will become available. Once you click on this button, it will gray out again until the next time you click on a file name.

2.4 Replace button

Use this button to open a file in the **Selected files** list. This button will be grayed out until you click on a file in the list and have already placed plots into the **Active plots list**. This button differs from the **Open button** in one significant way. If you have already opened a file and have created plots, you can use the **Replace** button to open the new file and place the new file's variables into the existing plots. This allows you to reuse existing plots, but only works if the new file contains variables of the same names. You must also make sure that the **Reset attributes on replace toggle button** in the **Preferences** window is off, unless you want all your plot attributes to reset to their defaults when you replace the file. Once you click on the **Replace** button, it will gray out again until the next time you click on a file name (and you have plots in the **Active plots list**).

2.5 Overlay button

Use this button to open a file in the **Selected files** list. This button will be grayed out until you click on a file in the list and have already placed plots into the **Active plots list**. This button is like the **Replace button**, but it differs from the **Replace button** in one significant way. If you have already opened a file and have created plots, you can use the **Overlay** button to open the new file and create copies of the existing plots, but have them use the new file's variables. This allows you to create new plots which are just like the existing plots, making comparisons much easier. However, this only works if the new file contains variables of the same names. Once you click on the **Overlay** button, it will gray out again until the next time you click on a file name (and you have plots in the **Active plots list**).

2.6 Output window icon



Figure 2-3: No new info

Pressing this icon brings up a window which has all kinds of information in it. Some types of information the window might display include the graphics driver used and pick and query output. Each time information is sent to the window, the icon changes from a blue “i” to a red “!” to inform you that new



Figure 2-4: New info

information has arrived. After you open the window, the icon reverts to the information icon to indicate you have already read the information and needn't look at the window again.

2.7 Output window

Various kinds of information are printed to this window, which pops up after you click on the **Output window icon**. One type of information is the type of renderer you are using (e.g., X11, GL, Software Renderer), which will be listed when you first bring up MeshTV. Warnings, errors, and informational messages are also echoed here, as well as being presented in a pop-up window. Results from certain operations, like the Pick & Query command, appear in this window.

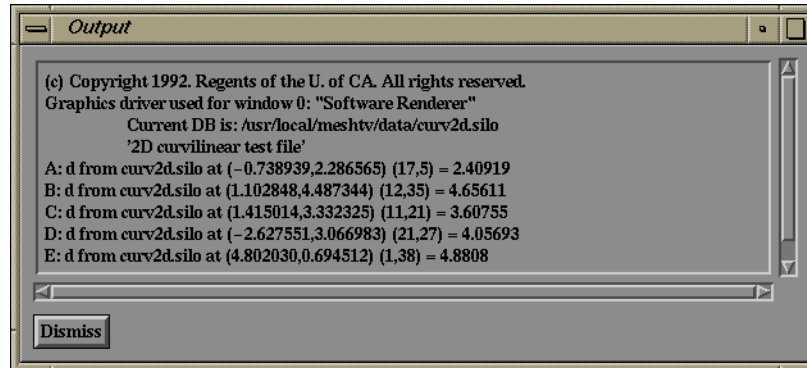


Figure 2-5: Output window

2.8 Busy clock icon

MeshTV informs you it's busy by placing a busy clock icon in this area. While you can still manipulate items in the MeshTV **Main** window, you should not expect to see results from your actions until the clock icon goes away.

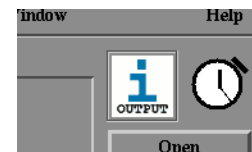


Figure 2-6: Busy clock

3.0 Animation area

3.1 Overview

This area, shown in Figure 2-7, provides file animation control. The panel consists of a slider scale, a text field, and five buttons. The leftmost button goes back one frame in the animation sequence, the next button plays the animation in reverse, the middle button stops the animation, the next button plays the animation, and the rightmost button moves forward one frame in the animation sequence. This control panel remains grayed out until you open a file that can be used in an animation.

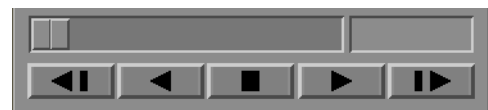


Figure 2-7: Animation area

MeshTV cycles through multiple SILO files to perform animations. These files must be in a “family,” which means they must follow a naming convention where the root name is the same, followed by increasing numbers. For example, the files “rect0001.silo” and “rect0002.silo” comprise a family of files. These two files could be animated by MeshTV.

3.2 Animation slider

This slider shows where you are in the animation. If the problem’s time steps are regular, 3/4 on the slider bar indicates both 3/4 in time through the problem run and 3/4 in distance (number of frames). If the problem’s time steps are irregular, then the slider bar will either indicate distance through the animation, or time through the animation, depending on how the file was opened. If file times were read in when the files were opened, the slider bar will indicate time, else it will indicate distance. You can specify whether file times are read in by pulling up the **Preferences** window from the **Extra** menu at the top of the MeshTV **Main** window. You want the **Ignore file times toggle button**. When you turn this off, MeshTV will read in the file times, else it will ignore them.

3.3 Animation slider text field

This text field lists either the current frame number or the time of the animation. You can type in your own number to jump to a particular frame or time. The number entered indicates distance through the animation or time through the animation, depending on how the file was opened. If file times were read in when the files were opened, the number indicates time, else it indicates distance. You can determine whether file times are read in by pulling up the **Preferences** window from the **Extra** menu at the top of the MeshTV **Main** window. You want the **Ignore file times toggle button**. When you turn this off, you will read in the file times, else you will ignore them.

3.4 Reverse button



Pressing this button moves the animation back one frame.

3.5 Back button



Pressing this button plays the animation in reverse.

3.6 Stop button



Pressing this button stops a running animation.

3.7 Play button



Pressing this button starts the animation.

3.8 Advance button



Pressing this button advances the animation by one frame.

4.0 Active plots area

4.1 Overview

MeshTV can have up to 16 visualization windows. This area, shown in Figure 2-8, allows you to select to which window you want to draw, whether to replace existing plots or let them overlay each other when you add a new one, and whether to have plot attributes automatically apply. It contains a list of active plots and a capability for adding and modifying plots via the **Plots**, **Plot attributes**, and **Variables** menus, which are described in Chapter 3. It also provides a method for performing operations on the plots via the **Operators** and **Operator attributes** menus, which are described in Chapter 4. You can also choose whether to apply operators to all plots or just the selected ones. Note that the list of active plots will be empty when MeshTV first starts.

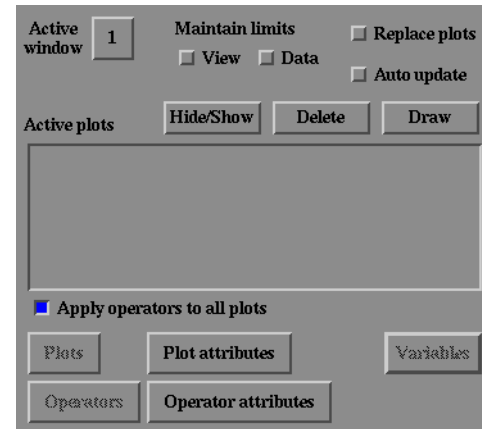


Figure 2-8: Active plots area

4.2 Active window

Figure 2-9 shows an example of the **Active window** pulldown when there are four visualization windows open. There will always be one number for every open window.

This pulldown allows you to select the visualization window to which you want to plot. To choose a window, pull down the menu and select the window number. That window becomes the active window, and any new plots will be drawn there.

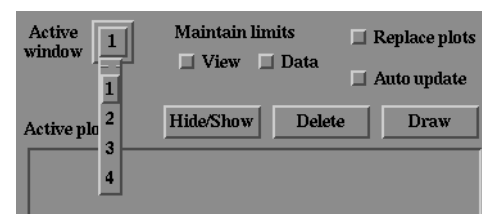


Figure 2-9: Active window pulldown

The dotted line at the top of the menu indicates that this is a “tear-off” menu, which means that if you click on the dotted line, the menu will “tear-off” and stay permanently posted until you close the window, or press the ESC key while in the window.

4.3 Maintain limits label and toggles

When the **Maintain limits** toggles are off, MeshTV rescales the view according to the limits of the plot and recalculates the data extents according to the full spectrum of data. If you wish to fix the limits of the view at the current plot size, select the **View** toggle. If you wish to fix the data extents so they remain the same when you switch variables or files (this keep pseudocolor plot colors from rescaling, for example) then select the **Data** toggle.

4.4 Replace plots toggle button

When **Replace plots** is off, which is the default, new plots are placed on top of existing plots. When **Replace plots** is on, a request for a new plot causes existing plots to be deleted when the new plot is drawn.

4.5 Auto update toggle button

When this option is turned on, any changes to plot attributes are applied automatically, so you don't need to press the **Apply** button in a window to apply the changes you have just made. This can be useful when you are just changing one attribute, but when you are changing multiple attributes it can really slow things down, since the plot is redrawn each time you make a change when this option is on.

4.6 Active plots list

This list contains the list of plots for a given visualization window. Each entry consists of the file number (which matches the file number in the **Selected files** list) followed by the plot type, like Boundary, the name of variable which was plotted, and any operators which have been applied, like Reflect. If you double click on a plot entry in this list, the plot attribute window for that plot entry will pop up. To see the plots for a different output window, you must make that window the active window. To set the active window, pull down the **Active window** menu and select the number of the window you want. The selected window becomes the active window, and the **Active plots** list updates with the plots for that window.

If you want to change multiple plots of the same type, select the plots you want to modify and then bring up the appropriate attributes window via the **Plot attributes** menu. Any changes you make and apply via that window will apply to all selected plots of that type.

4.7 Delete button

To remove a plot from the **Active plots** list, click on the plot in the list and then press the **Delete** button. You can also select multiple plots and use this button to delete them.

4.8 Draw button

Press the **Draw** button when you want to draw the plots in the **Active plots** list.

Note that if the **Reset attributes on replace toggle button** in the **Preferences** window is on, pressing the **Draw** button without adding a plot causes plots to be redrawn with the attributes reset. The **Reset attributes on replace toggle button** is off by default.

4.9 Hide/Show button

To hide a plot that has already been drawn, click on the plot in the **Active plots** list and then press the **Hide/Show** button. To unhide the plot, select the plot and press the **Hide/Show** button again. You can also select multiple plots and use this button to hide or show them.

4.10 Apply operators to all plots toggle button

Operators (like reflect, for example) only apply to highlighted plots unless this option is activated. If this option is on, chosen operators will be applied to all plots in the **Active plots** list in the MeshTV **Main** window.

5.0 Notepad area

5.1 Overview

The **Notepad** area allows you to post windows to one location. Unless you have posted windows and then selected **Save settings** from the **File** menu at the top of the MeshTV **Main** window, this area displays only the **Command Line Interface** window. Figure 2-10 also shows the **Animation** window posted to the area to provide a more interesting example.

All windows which have a **Post** button can be placed into the **Notepad** by pressing the **Post** button. Once a window is posted, its **Post** button switches to an **Unpost** button.

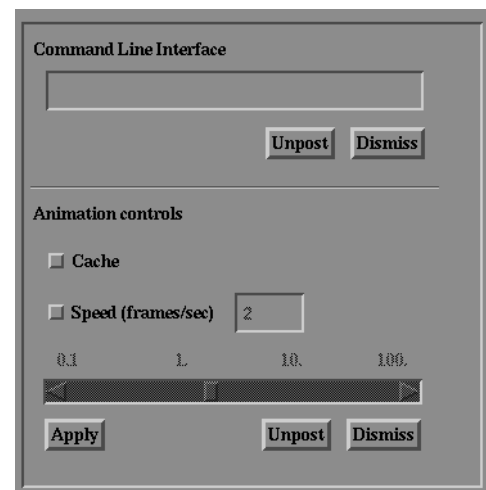


Figure 2-10: Notepad area

Chapter 3

Plots, Plot Attributes, and Variables

1.0 Overview

This chapter covers items found under the **Plots**, **Plot attributes**, and **Variables** menus in the **Active plots** area of the MeshTV **Main** window. The pulldown menus are shown at the bottom of Figure 3-1.

This section gives a quick description for each menu item. More detailed descriptions follow.

Plots

The **Plots** pulldown menu allows you to select different plot types, like Boundary or Mesh, using the variables in the file.

Plot attributes

The **Plot attributes** pulldown menu allows you to pop up windows which let you set attributes for each plot.

Variables

The **Variables** pulldown menu presents alternate variables for existing plots. To change variables, select one or more plots, then choose a new variable from the list.

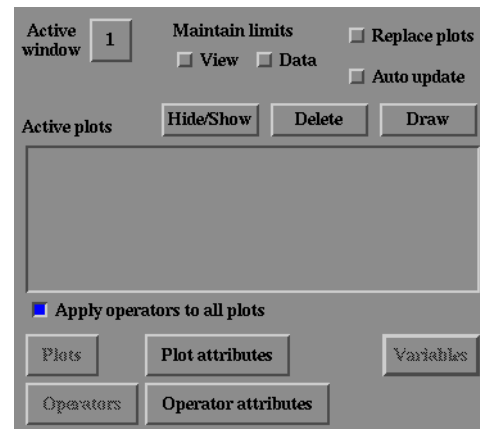


Figure 3-1: Active plots area

2.0 Plots menu

2.1 Overview

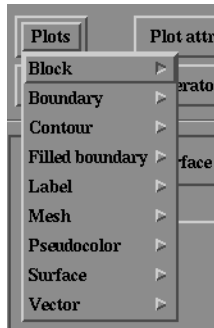


Figure 3-2: Plots menu

Selecting the plots menu brings up a list of MeshTV plot types. Plots for which your data file has no data will be grayed out. If you can display a plot, pulling the mouse toward the right while holding down the left button will show you which variables are available. Release the mouse button when your cursor is over the variable you want to plot, and a new plot will appear in the **Active plots** list. Note that this menu will be grayed out (and inaccessible) until you open a file.

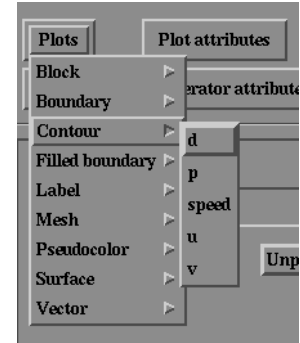


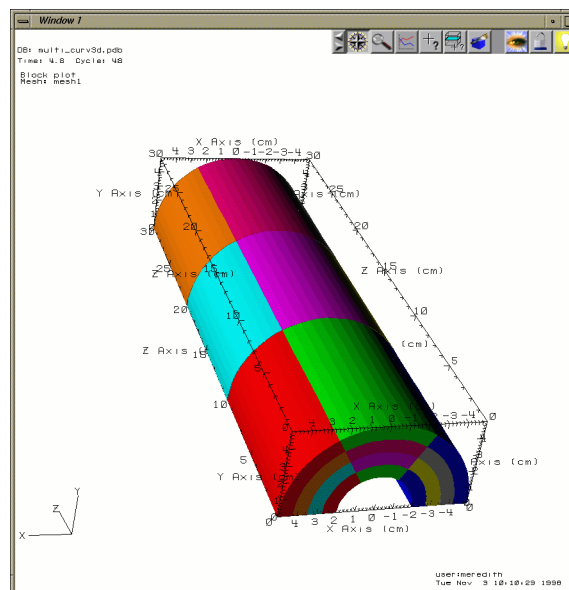
Figure 3-3: Contour variables

Available plot types are **Block**, **Boundary**, **Contour**, **Filled Boundary**, **Label**, **Mesh**, **Pseudocolor**, **Surface**, and **Vector**.

In the example shown in Figure 3-2, if you pulled to the right on the **Contour** entry, you would see a list of variables as in Figure 3-3, assuming you first opened the MeshTV sample data file, *curv2d.silo*.

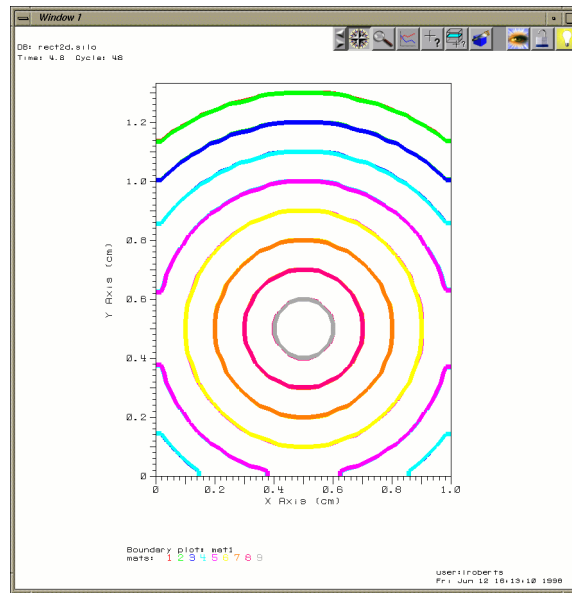
2.2 Block plot

This plot displays a color-coded breakdown of the blocks, or domains, in a multiblock mesh. With the fill option off, it displays the boundaries between these blocks.



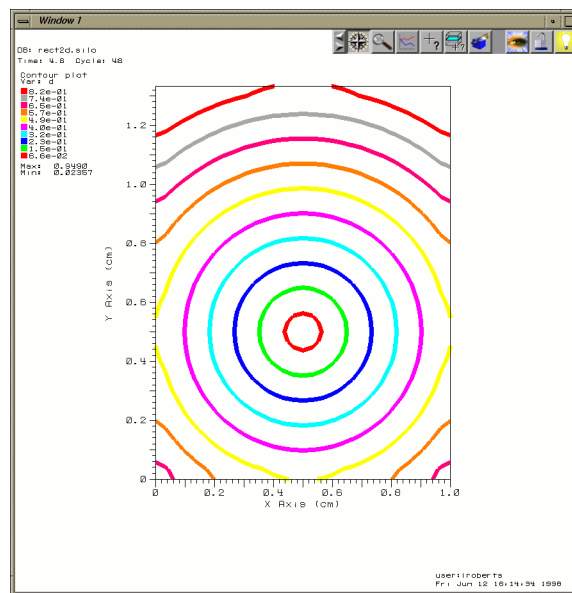
2.3 Boundary plot

This plot displays the boundaries between materials.



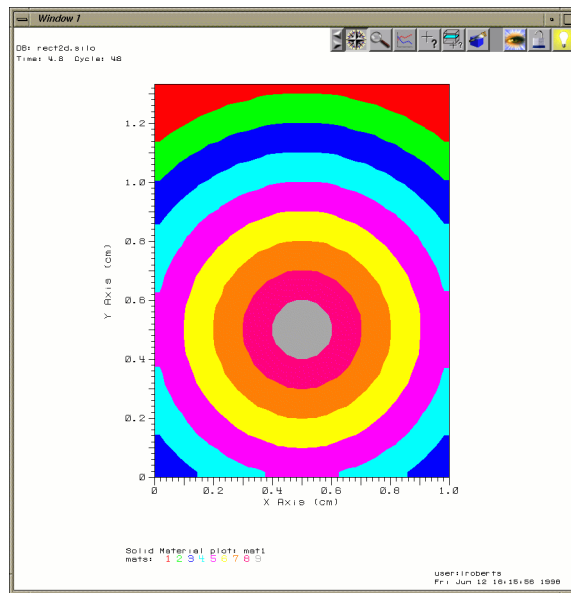
2.4 Contour plot

This plot displays the location of values for scalar variables like density or pressure, using lines for 2D plots and surfaces for 3D plots. In visualization terms, these plots are isosurfaces.



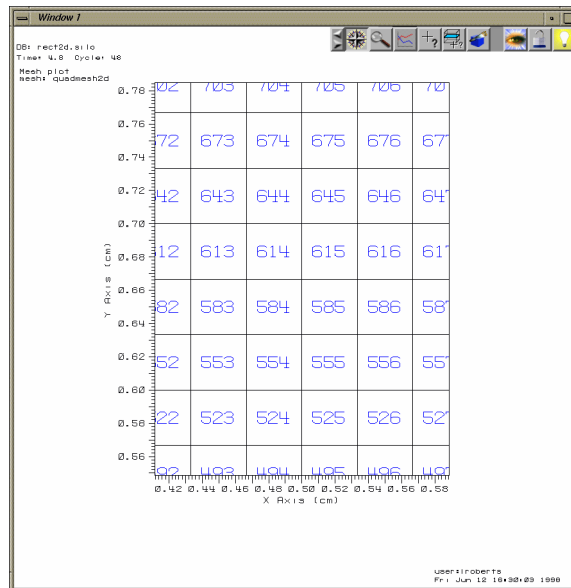
2.5 Filled boundary plot

This plot displays a different color for each material in the problem.



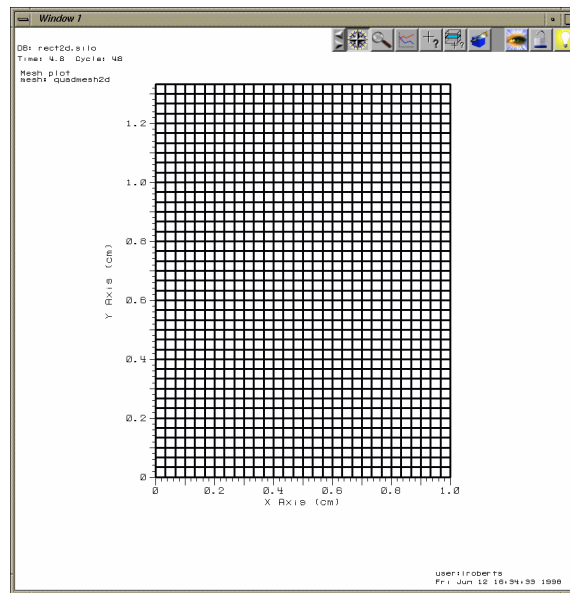
2.6 Label plot

This plot assigns numbers or strings to nodes and/or zones in the problem.



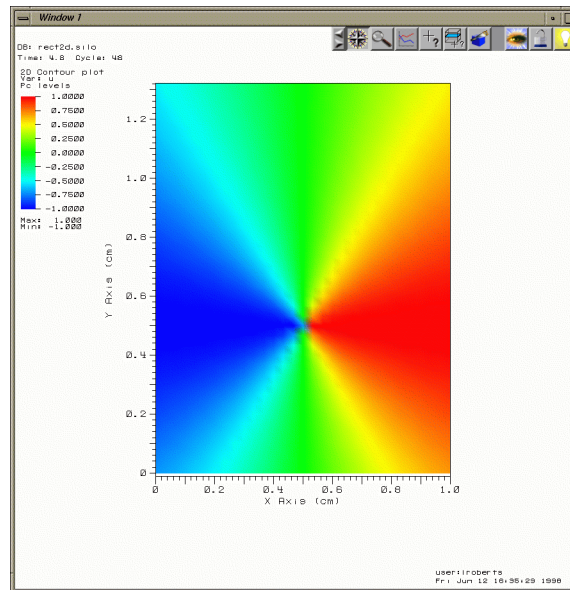
2.7 Mesh plot

This displays the problem mesh.



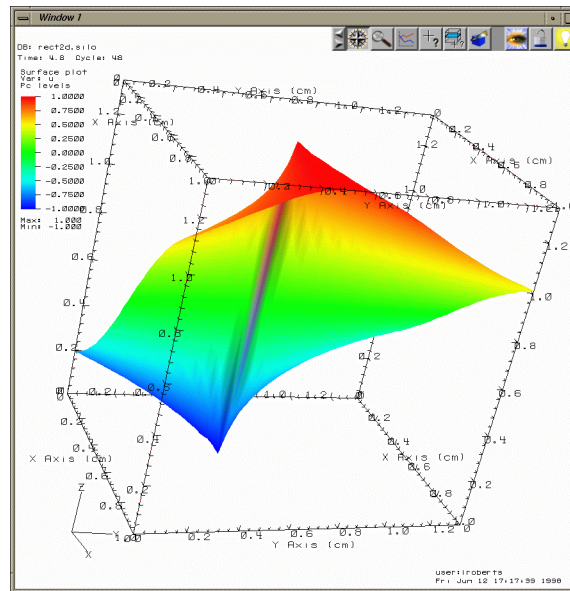
2.8 Pseudocolor plot

This plot uses color to show variable values in a problem.



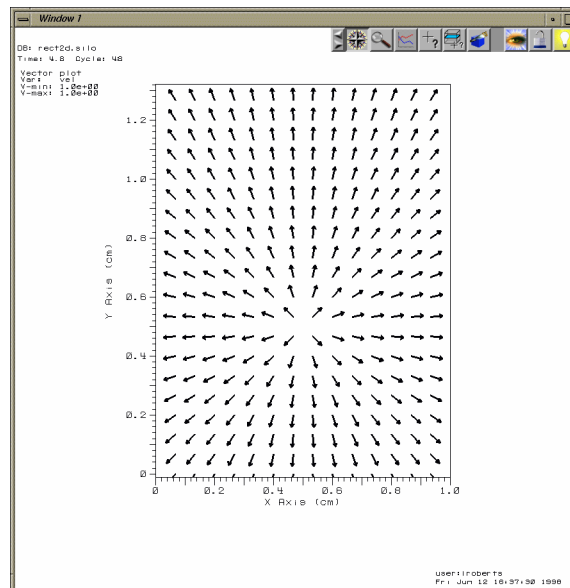
2.9 Surface plot

This plot displays 2D information as a surface, using the value of the given variable for the height.



2.10 Vector plot

This plot displays vector variables.



3.0 Plot attributes menu

3.1 Overview

This pulldown menu provides access to windows that allow you to change plot attributes. Menu items with a ... in the name indicate that a window will open when the item is selected. The dotted line at the top of the menu denotes a "tear-off" menu, which means that when you click on the dotted line, the menu will "tear off" and stay posted until you close the window or press the ESC button while the window is selected.

The available options are **Block**, **Boundary**, **Contour**, **Curve**, **Filled boundary**, **Label**, **Mesh**, **Pick**, **Pseudocolor**, **Refine**, **Surface**, and **Vector**. Each window allows you to set options for the plot type of the same name. The attributes can be applied to existing plots, or they can become the default for any plots created after changing the attributes.

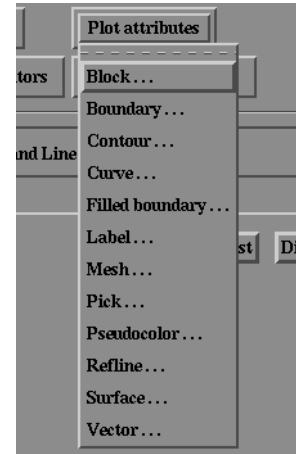


Figure 3-4: Plot attributes

4.0 Similar items

Many plots share similar attributes, like the ability to create log plots, as well as similar widgets, like the **Apply** and **Dismiss** buttons. Rather than repeat the descriptions for these elements in the description for each window, common elements are described here, beginning with a table which details which windows include which elements, and ending with a description of the elements themselves.

MeshTV Plots	Apply	Dismiss	Fill Type	Legend	Line Color	Line Style	Line Width	Log	Make Default	Multiple	Post	Reset	Single
Block	•	•	•	•		•	•		•	•	•	•	•
Boundary	•	•		•		•	•		•	•	•	•	•
Contour	•	•		•		•	•		•	•	•	•	•
Curve	•	•		•	•	•	•		•		•	•	
Filled Boundary	•	•		•					•		•	•	
Label	•	•		•					•		•	•	

MeshTV Plots	Apply	Dismiss	Fill Type	Legend	Line Color	Line Style	Line Width	Log	Make Default	Multiple	Post	Reset	Single
Mesh	
Pick	.	.									.		
Pseudo-color	
Reference Line	
Surface	
Vector	

4.1 Apply button

Pressing the **Apply** button applies changes you made in the window containing the button. If you don't want your changes applied, press the **Dismiss** button without pressing **Apply**.

4.2 Dismiss button

When you wish to close a window, press the **Dismiss** button. You should press the **Apply** button before you press the **Dismiss** button if you want your changes applied. If you don't want to apply your changes, pressing only the **Dismiss** button (without pressing **Apply**) will return you to previously applied values.

4.3 Fill type radio buttons

These buttons determine the type of fill used when drawing a plot. **Filled** means solid, filled, and shaded (in three dimensions). **Internal** means only the boundary between objects is drawn. **Wireframe** means only the outline is drawn.

4.4 Legend toggle button

This button determines whether a legend is included with the plot. So long as legends for plots are globally turned on (via a control in the **Annotation** window), a legend is displayed for a given plot when this option is on.

4.5 Line color button

This button allows you to select a new color for your lines using the **Palette editor**. Clicking the color button will activate the **Palette editor** so a new line color can be selected. The number displayed on the line color button represents the index of the color that is assigned to the line color.

4.6 Line style menu

This menu allows you to select one of four line styles. These are solid, dashed, dotted, and dot-dashed.

4.7 Line width menu

This menu allows you to select one of four line widths: hair width, narrow, medium, and wide.

4.8 Log button

This option determines whether data and the accompanying legend are displayed in a linear or log format. If legends are turned on, the legend and data are displayed with a log scale when this option is on.

4.9 Make default button

If you select this button, changes you have made become the default for future plots of this type. Existing plots are unaffected, and these defaults last only for the duration of the MeshTV session. To permanently save the new defaults, you must select **Save settings** from the **File** menu after setting them with the **Make default** button.

4.10 Multiple button

This button allows you to use multiple colors in a plot. If you select this button, the selected plot will use multiple colors instead of a single color.

4.11 Post button

If you plan to use a window repeatedly, but you don't want it cluttering up your screen, you might want to post it. When a window is posted, it appears in the **Notepad** area at the bottom of the MeshTV **Main** window, where it remains until it is unposted. Press this button to post the window. Once the window is posted, this button will change to **Unpost**.

A posted window will only appear in the **Notepad** for the duration of the current MeshTV session. To permanently retain a window in the Notepad, select **Save settings** from the **File** menu.

4.12 Reset button

To restore a window to default settings, undoing any changes you have made, press the **Reset** button. To apply these default values to selected plots, press the **Apply** button. Default values are the ones in effect when you started MeshTV, unless the window has a **Make default button**, in which case the defaults will be the values in effect when you pressed this button, if you did.

4.13 Single button

This button allows you to use a single color for the entire plot. If you select this button, the selected plot will use a single color instead of multiple colors. This button is often accompanied by another button that allows you to select the single color with the **Palette editor**. Clicking the companion button will activate the **Palette editor** so a new single color can be selected. The number displayed on the companion button represents the index of the single color.

5.0 Block plot attributes

5.1 Block plot attributes menu item

This item pops up a window that allows you to set plot attributes for block plots. If you select a block plot from the **Active plots** list in the MeshTV **Main** window, then attributes you change and apply will be applied to the selected plot. Unselected block plots will not be changed. If you select this item without selecting a block plot first, you will have to select a block plot before you can apply the changes. Note that if you make any changes in this window and then select a block plot, the attributes of the selected plot will replace your changes.

You can modify the default settings for attributes of future plots by opening this window, making your changes, and then pressing the **Make default** button. This affects only future plots, not existing ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

5.2 Block plot attributes window

The **Block plot attributes** window looks like Figure 3-5, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing or future block plots. The attributes include things like line width, style, and color, and the ability to change the fill type.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

5.3 Point size text field

When viewing a point mesh, the size of the points can be adjusted using this text field. The default point size is 0.50.

5.4 Separate toggle button

When viewing a mesh using type **Internal** or type **Wireframe**, often the lines or polygons can overlap each other, making it more difficult to determine the true color to which each block corresponds. Use this toggle button to separate each block by a small amount and prevent the blocks from overlapping.

6.0 Boundary plot attributes

6.1 Boundary plot attributes menu item

This item pops up a window that allows you to set plot attributes for boundary plots. If you select a boundary plot from the **Active plots** list in the MeshTV **Main** window, then attributes you change and apply will be applied to the selected plot. Unselected boundary plots will not be changed. If you select this item without selecting a boundary plot first, you will have to select a boundary plot before you can apply the changes. Note that if you make any changes in this window and then select a boundary plot, the attributes of the selected plot will replace your changes.

You can modify the default settings for attributes of future plots by opening this window, making your changes, and then pressing the **Make default** button. This affects only

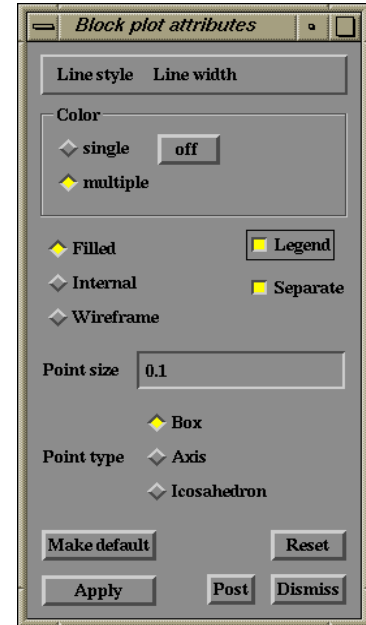


Figure 3-5: Block plot attributes

future plots, not existing ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

6.2 Boundary plot attributes window

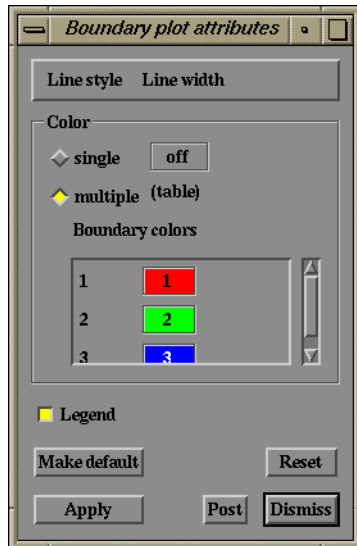


Figure 3-6: Boundary plot attributes

The **Boundary plot attributes** window looks like Figure 3-6, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing or future boundary plots. The attributes include things like boundary color and volume fraction.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

6.3 Boundary color buttons

These color buttons display the colors for each boundary. To change boundary colors, click on a boundary color button to activate the **Palette editor**.

Selecting a new color for a boundary in the **Palette editor** will update the boundary color in this window. The number displayed on a boundary color button represents the index of the color that is currently assigned to the given boundary.

7.0 Contour plot attributes

7.1 Contour plot attributes menu item

This item pops up a window that allows you to set plot attributes for contour plots. If you select a contour plot from the **Active plots** list in the MeshTV **Main** window, then attributes you change and apply will be applied to the selected plot. Unselected contour plots will not be changed. If you select this item without selecting a contour plot first, you will have to select a contour plot before you can apply the changes. Note that if you make any changes and then select a contour plot, you will lose your changes since the window will update with attributes from the selected plot.

You can modify the default settings for attributes of future plots by opening this window, making your changes, and then pressing the **Make default** button. This affects only future plots, not existing ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

7.2 Contour plot attributes window

The **Contour plot attributes** window looks like Figure 3-7, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing or future contour plots. The attributes include things like contour color and whether a legend should be plotted.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

7.3 Contour color buttons

These color buttons display the colors for each contour. To change contour colors, click on a contour color button to activate the **Palette editor**. Selecting a new color for a contour in the **Palette editor** will update the contour color in this window. The number displayed on a contour color button represents the index of the color that is currently assigned to the given contour.

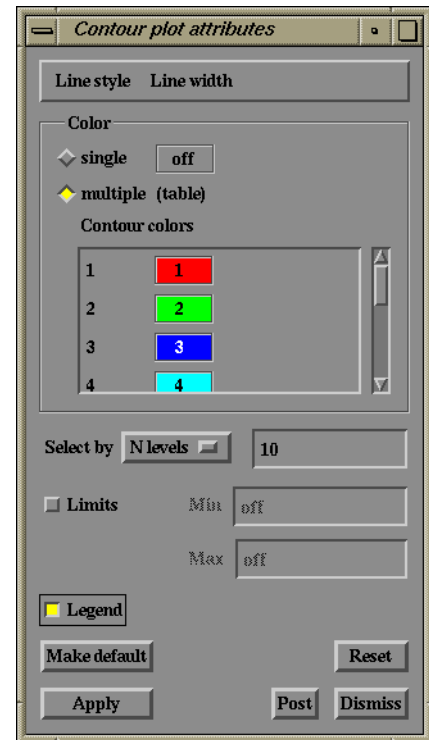


Figure 3-7: Contour plot attributes

7.4 Select by options

By default, MeshTV constructs 10 levels into which the data fall. These levels are linearly interpolated between the data minimum and data maximum. However, you can set your own number of levels, specify the levels you want to see, or indicate percentages for the levels.

Three options exist for specifying how levels are determined. These are **N levels**, **Levels**, and **Percent**. **N levels**, the default method, allows you to specify the number of levels which will be generated, with 10 being the MeshTV default. **Levels** requires you to specify real numbers for the levels you want to see. **Percent** takes a list of percentages, like 50.5, 55, 60, and 75. Using the numbers just mentioned, the first contour would be placed at the value which is 50.5% of the way between the minimum and maximum data values. The next contour would be placed at the value which is 55 percent of the way between the minimum and maximum data values, and so forth.

The **N levels** and **Percent** selection options also allow you to select the data range between which to interpolate the levels, if you select the **Limits** toggle button. When the **Limits** toggle is on, you can specify a minimum and/or maximum value, and the interpolation algorithm will select levels based linearly between those values, though all data will be displayed.

7.5 Limits toggle button

If you want to restrict the range of data used by the contour plot to generate its levels, this is the option to use. Once you select the button, the **Min** and **Max** text fields become active, and you can enter in the range to use. You can change one or both fields.

This option is only active when the **Select by** option is set to either **N levels** or **Percent**.

7.6 Min text field

This text field becomes active once the **Limits** toggle button is selected. When the field contains the string “off,” the actual data minimum is used, else the contour plot uses the number entered in the text field as the minimum number for calculating where contour levels are placed. Contour levels are always placed between the minimum and maximum values.

7.7 Max text field

This text field becomes active once the **Limits** toggle button is selected. When the field contains the string “off,” the actual data maximum is used, else the contour plot uses the number entered in the text field as the maximum number for calculating where contour levels are placed. Contour levels are always placed between the minimum and maximum values.

8.0 Curve plot attributes

8.1 Curve plot attributes menu item

This item pops up a window that allows you to set plot attributes for curve plots. If you select a curve plot from the **Active plots** list in the MeshTV **Main** window, then attributes you change and apply will be applied to the selected plot. Unselected curve plots will not be changed. If you select this item without selecting a curve plot first, you will have to select a curve plot before you can apply the changes. Note that if you make any changes and then select a curve plot, you will lose your changes since the window will update with the attributes of the selected plot.

You can modify the default settings for attributes of future plots by opening this window, making your changes, and then pressing the **Make default** button. This affects only future plots, not existing ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

8.2 Curve plot attributes window

The **Curve plot attributes** window looks like Figure 3-8, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing and future curve plots. The attributes include things like line color and whether a legend should be plotted. To assign the output window which receives curve plots (when you draw lines in the line-out mode) or to indicate that you want to replace existing plots when a new curve is generated, you must open the **Line-out** window in the **Controls** menu at the top of the MeshTV **Main** window.

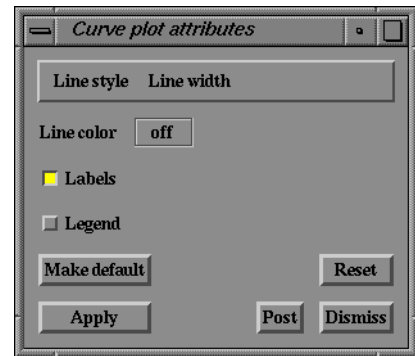


Figure 3-8: Curve plot attributes

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

8.3 Labels toggle

This toggle allows you to turn curve labels on and off. Labels provide identification, but if you have too many curves in a given window, the labels can obscure too much information to be useful.

9.0 Filled boundary plot attributes

9.1 Filled boundary plot attributes menu item

This item pops up a window that allows you to set plot attributes for filled boundary plots. If you select a filled boundary plot from the **Active plots** list in the MeshTV **Main** window, then attributes you change and apply will be applied to the selected plot. Unselected filled boundary plots will not be changed. If you select this item without selecting a filled boundary plot first, you will have to select a filled boundary plot before you can apply the changes. Note that if you make any changes and then select a filled boundary plot, you will lose your changes since the window will update with the attributes of the selected plot.

You can modify the default settings for attributes of future plots by opening this window, making your changes, and then pressing the **Make default** button. This affects only future plots, not existing ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

9.2 Filled Boundary plot attributes window

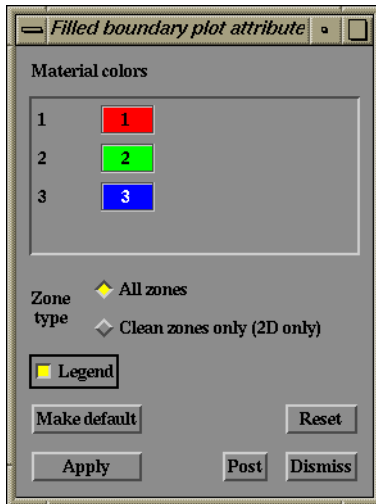


Figure 3-9: Filled boundary attribute

The **Filled boundary plot attributes** window looks like Figure 3-9, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing or future filled boundary plots. The attributes include things like zone type, volume fraction, and whether to plot the legend.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

9.3 Material color buttons

These color buttons display the colors for each material. To change material colors, click on a material color button to activate the **Palette editor**. Selecting a new color for a material in the **Palette editor** will update the material color in this window. The number displayed on a material color button represents the index of the color that is currently assigned to the given material.

9.4 Zone type radio buttons

These radio buttons determine if all zones will be included in the filled boundary plot, or if only clean zones will be included. "Clean" zones contain only one material as opposed to two or more. Zones with two or more materials are called "mixed" zones.

10.0 Label plot attributes

10.1 Label plot attributes menu item

This item pops up a window that allows you to set plot attributes for label plots. If you select a label plot from the **Active plots** list in the MeshTV **Main** window, then attributes you change and apply will be applied to the selected plot. Unselected label plots

will not be changed. If you select this item without selecting a label plot first, you will have to select a label plot before you can apply the changes. Note that if you make any changes and then select a label plot, you will lose your changes since the window will update with the attributes of the selected plot. You can modify the default settings for attributes of future plots by opening this window, making your changes, and then pressing the **Make default** button. This affects only future plots, not existing ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

10.2 Label plot attributes window

The **Label plot attributes** window looks like Figure 3-10, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing or future label plots. The attributes include things like labeling zones, nodes, or both; how many labels to show; where labels should be placed; and how large the labels should be.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

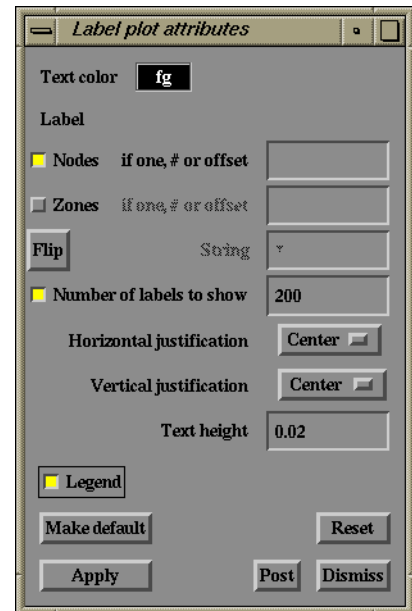


Figure 3-10: Label plot attributes

10.3 Nodes toggle button

When this toggle is selected, nodes of the mesh are labeled. Unless a specific label is specified, all nodes are labeled. If a value is specified in the **Node offset** text field, then only that node is labeled. When the **Nodes** toggle is off, the **Node offset** text field and accompanying label become insensitive to user input.

10.4 Node offset label and text field

When the **Nodes** toggle button is selected, this label is no longer grayed out, and you can type values into the text field. If you type a value into the text field, MeshTV finds the node represented by that value and labels it with the contents of the **String** text field. As soon as you type a value into the **Node offset** text field, the **String** text field and accompanying label become active, allowing you to specify the string you wish to use for the label.

Valid values are an integer number for a given node, or i,j (for 2D) and i,j,k (for 3D) locations. The location notation only works with logically rectangular meshes, so unstructured meshes do not allow this kind of input.

10.5 Zones toggle button

When this toggle is selected, zones of the mesh are labeled. Unless a specific label is specified, all zones are labeled. If a value is specified in the **Zone offset** text field, then only that zone is labeled. When the **Zones** toggle is off, the **Zone offset** text field and accompanying label become insensitive to user input.

10.6 Zone offset label and text field

When the **Zones** toggle button is selected, this label is no longer grayed out, and you can type values into the text field. If you type a value into the text field, MeshTV finds the zone represented by that value and labels it with the contents of the **String** text field. As soon as you type a value into the **Zone offset** text field, the **String** text field and accompanying label become active, allowing you to specify the string you wish to use for the label.

Valid values are an integer number for a given zone, or i, j (for 2D) and i, j, k (for 3D) locations. The location notation only works with logically rectangular meshes, so unstructured meshes do not allow this kind of input.

10.7 Flip button

The **Flip** button allows you to easily switch between labeling nodes or zones with just the push of a button. This button is active only when either the **Nodes** or **Zones** toggle is active, and when pressed, it deselects the selected type (Node or Zone) and selects the other type.

10.8 String label and text field

When you are labeling a single node or zone, you can indicate that a particular string, instead of the standard node/zone number, be placed at that location. The string might be something like “*”, or even “Here”, though one character strings usually work better due to space constraints. The **String** text field is insensitive to user input and the label is grayed out until only one node and/or zone is being labelled.

10.9 Number of labels to show toggle and text field

When this toggle is turned on, MeshTV displays the number of labels indicated in the accompanying text field, else MeshTV displays all labels. By specifying a lower number,

like 200, you can reduce the drawing time for many label plots and possibly make things more readable since labels will be less likely to overlap, but you might not get the label you want. However, if you increase the number too much, the labels can blanket the object, obscuring both themselves and any other plots. Note that this number indicates the number which will be visible in the picture, so if you zoom in, you will see labels “moving” as new labels are “turned on.” If you zoom in closely enough, all the labels will be drawn.

10.10 Horizontal justification option menu

Use this option to control the horizontal placement of a label. The label can be centered horizontally on the node/zone, or placed to the left or right of the node/zone. The default is to center the label.



Figure 3-11: Horizontal justification menu

10.11 Vertical justification option menu

Use this option to control the vertical placement of a label. The label can be centered vertically on the node/zone, or placed above (Top) or below (Bottom) the node/zone. The default is to center the label.

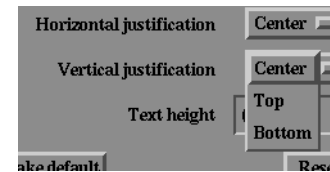


Figure 3-12: Vertical justification menu

10.12 Text height text field

Use this button to type in text heights for the labels. This particular option might require a little experimenting before you find the height you like. The default value is 0.02.

11.0 Mesh plot attributes

11.1 Mesh plot attributes menu item

This item pops up a window that allows you to set plot attributes for mesh plots. If you select a mesh plot from the **Active plots** list in the MeshTV **Main** window, then attributes you change and apply will be applied to the selected plot. Unselected mesh plots will be unchanged. If you select this item without selecting a mesh plot first, you will have to select a mesh plot before you can apply the changes. Note that if you make any changes and then select a mesh plot, you will lose your changes since the window will update with the attributes of the selected plot.

You can modify the default settings for attributes of future plots by opening this window, making your changes, and then pressing the **Make default** button. This affects only future plots, not existing ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

11.2 Mesh plot attributes window

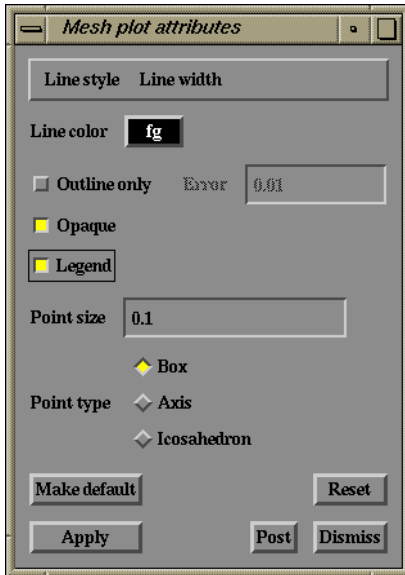


Figure 3-13: Mesh plot attributes

The **Mesh plot attributes** window looks like Figure 3-13, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing or future mesh plots. The attributes include things like whether the legend will be shown and how much of the mesh to draw.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

11.3 Outline only toggle

When this toggle is selected, a subset of the mesh is drawn, sometimes drawing only the outline. How much of the mesh is drawn is set by the **Error** text field,

which is grayed out until this toggle is activated.

While this works for both 2D and 3D data files, using this option yields an outline only for 2D data sets.

11.4 Error text field

This text field becomes active once you turn on the **Outline only** toggle. The numeric value in this field ranges from 0.0 to 1.0, where 0.0 means draw the entire mesh and 1.0 means remove as much as possible. This number actually controls how much of the mesh is drawn by signifying how much deviation (error) from a straight line or surface is allowed before lines/surfaces will be combined. Thus, if the allowed deviation is 0, then all lines must be drawn. If the allowed deviation is 0.5, then lines can have a significant angle and still be collapsed into one line. Therefore, for some meshes and all 2D meshes, the only real options are either 0.0 or 1.0. The default is 0.01.

11.5 Point size

When plotting a point mesh, you can set the size of the points. The default size is 0.05.

12.0 Pick plot attributes

12.1 Pick plot attributes menu item

This item pops up a window that allows you to set the text color for future pick plots. Once you have performed the pick action, you can no longer change the text color of the marker letter it produces, so you need to set the text color first.

You can modify the default settings for attributes of future plots by opening this window, making your changes, and then pressing the **Make default** button. This affects only future plots, not existing ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

12.2 Pick plot attributes window

The **Pick plot attributes** window looks like Figure 3-14, if you are using MeshTV's defaults. If you aren't, the value of the text color might be different.

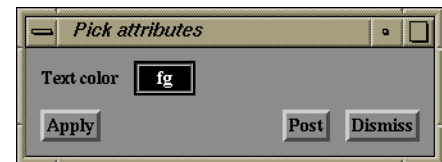


Figure 3-14: Pick plot attributes

This window allows you to set the text color used in future pick plots.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

12.3 Text color button

This button allows you to select a new color for your text using the **Palette editor**. Click the button to activate the **Palette editor** and select a new text color.

13.0 Pseudocolor plot attributes

13.1 Pseudocolor plot attributes menu item

This item pops up a window that allows you to set plot attributes for pseudocolor plots. If you select a pseudocolor plot from the **Active plots** list in the MeshTV **Main** window, then attributes you change and apply will be applied to the selected plot. Unselected pseudocolor plots will not be changed. If you select this item without selecting a pseudocolor plot first, you will have to select a pseudocolor plot before you can apply the changes. Note that if you make any changes and then select a pseudocolor plot, you will lose your changes since the window will update with the attributes of the selected plot.

You can modify the default settings for attributes of future plots by opening this window, making your changes, and then pressing the **Make default** button. This affects only future plots, not existing ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

13.2 Pseudocolor plot attributes window

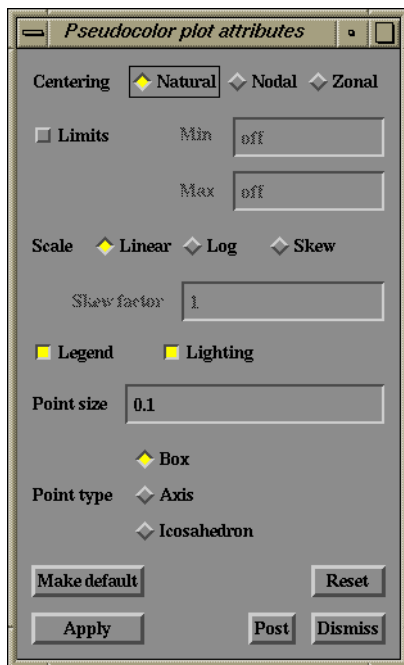


Figure 3-15: Pseudocolor attributes

The **Pseudocolor plot attributes** window looks like Figure 3-15, if you are using MeshTV’s defaults. If you aren’t, the values might be different.

This window allows you to set the attributes assigned to existing or future pseudocolor plots. The attributes include things like whether the legend will be shown, whether to display the data as nodal, zonal, or with its natural centering, and which data limits to use when making the plot.

If you don’t see a description for an item in this window, check in the Shared items section near the front of the chapter.

13.3 Centering toggles

These toggles determine how data in the pseudocolor plot is shown. **Natural** indicates to display the data with its natural centering. This means that node-centered data will be displayed at the nodes with colors being linearly interpolated between the nodes, and zone-centered data will be displayed as zonal values, giving a slightly “blocky” look to the picture. If you select **nodal**, then all information will be displayed at the nodes with color interpolated between. This will produce a smoother picture, but for variables which are actually zone-centered, you will lose some data (local minima and maxima). If you select **zonal**, all data will be displayed as if they were zone-centered. This will produce a blockier picture. Again, this will blur out minima/maxima for node-centered data.

13.4 Limits toggle

If you want to restrict the range of data used by the pseudocolor plot to generate its output, this is the option to use. Once you select this button, the **Min** and **Max** text fields will become active, and you can enter in the range to use. You can change one or both fields.

13.5 Min text field

This text field becomes active once the **Limits** toggle button is selected. When the field contains the string “off,” the actual data minimum is used, else the pseudocolor plot uses the number entered in the text field as the minimum number. When a user-supplied number is used, all data values below this minimum map to the same color as the user-supplied minimum.

13.6 Max text field

This text field becomes active once the **Limits** toggle button is selected. When the field contains the string “off,” the actual data maximum is used, else the pseudocolor plot uses the number entered in the text field as the maximum number. When a user-supplied number is used, all data values above this maximum map to the same color as the user-supplied maximum.

13.7 Scale radio buttons

These radio buttons determine the type of scaling used to display the data in the pseudocolor plot. There are three types of scaling: **linear**, **log**, and **skew**. Linear, which is the default, uses a linear mapping of data values to color values. Log scaling is used to map small ranges of data to larger ranges of color. Skew scaling goes one step further by using an exponential function based on a skew factor to adjust the mapping of data to color. The function used in skew scaling is $(s^d - 1) / (s - 1)$ where **s** is a scale factor greater than zero and **d** is a data value that has been mapped into a range from zero to one. You control how data is mapped to colors by changing the skew factor. A skew factor of one is equivalent to linear scaling but values either smaller or larger than one produce curves that will map either the high or low end of the data to a larger color range.

13.8 Skew factor text field

This text field becomes active once the pseudocolor scale has been set to skew. The skew factor text field is used to specify the skew factor used during skew scaling. Values used for the skew factor are real numbers greater than zero. A skew factor of 1 is equivalent to linear scaling while values like 0.1, 100 will map the high or low pseudocolor plot data to larger color ranges in order to better highlight differences in data values. When linear or log scaling are being used, this text field is disabled.

13.9 Lighting toggle

This toggle turns lighting on/off for the current pseudocolor plot. Note that this only affects 3D plots. Changing the value of this attribute will not change the appearance of a 2D plot. Lighting calculations significantly increase the time it takes to render a scene, so if speed is a necessity, then it might be best to turn lighting off. However, lighting adds

detail and depth to a pseudocolor plot, two characteristics important for movies. If your goal is to produce a movie file from a sequence of pseudocolor plots, then lit pseudocolor plots will probably look the best.

13.10 Point size text field

When creating a pseudocolor plot of a point mesh, you can set the point size. The default size is 0.05.

14.0 Reference line plot attributes

14.1 Reference line plot attributes menu item

This item pops up a window that allows you to set plot attributes for reference line plots. If you select a reference line plot from the **Active plots** list in the MeshTV **Main** window, then attributes you change and apply will be applied to the selected plot. Unselected reference line plots will not be changed. If you select this item without selecting a reference line plot first, you will have to select a reference line plot before you can apply the changes. Note that if you make any changes and then select a reference line plot, you will lose your changes since the window will update with the attributes of the selected plot.

You can modify the default settings for attributes of future plots by opening this window, making your changes, and then pressing the **Make default** button. This affects only future plots, not existing ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

14.2 Reference line plot attributes window

The **Reference line plot attributes** window looks like Figure 3-16, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing or future reference line plots. The attributes include things like setting the endpoints which define the line, and setting the line's color, style, and width attributes.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

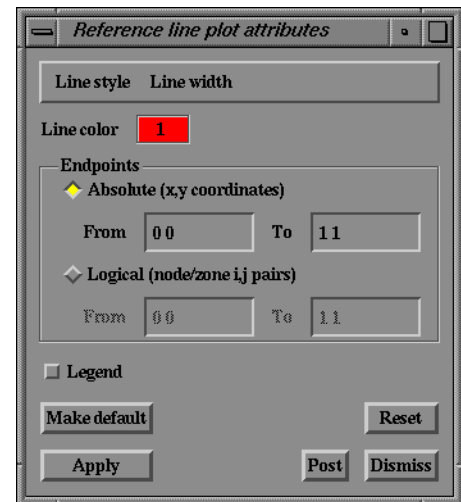


Figure 3-16: Reference line plot attributes

14.3 Endpoints area

This area allows you to change the endpoints of existing reference line plots. You can specify whether the line should be absolute (a straight line from the starting point to the ending point) or logical (a line which follows the mesh.)

14.3.1 Absolute endpoints radio button

When you switch a graphics output window into line-out mode (holding down the right mouse button and selecting **Mode** and **Line-out** from the pop-up menu), you can draw lines in the window.

Two kinds of lines can be drawn. By default, straight lines with absolute endpoints are drawn, though you can switch to drawing logical lines that follow the mesh by selecting the **Logical** toggle in the **Line-out** window found under the **Controls** menu.

When you select a reference line in the **Active plots** area, you can modify its endpoints by selecting the **Absolute endpoints** radio button and putting the coordinates of the starting point into the **From** text field and the coordinates of the ending point into the **To** text field. After you hit **Apply**, the endpoints of the line will snap to the points you entered. Note that this option applies only to existing lines, so the setting is not saved when you hit the **Make default** button.

14.3.2 Logical endpoints radio button

When you switch a graphics output window into line-out mode (holding down the right mouse button and selecting **Mode** and **Line-out** from the pop-up menu), you can draw lines in the window.

Two kinds of lines can be drawn. By default, straight lines with absolute endpoints are drawn, though you can switch to drawing logical lines that follow the mesh by selecting the **Logical** toggle in the **Line-out** window found under the **Controls** menu.

When you select a reference line in the **Active plots** area, you can modify its endpoints by selecting the **Logical endpoints** radio button and putting the coordinates of the starting point into the **From** text field and the coordinates of the ending point into the **To** text field. After you hit **Apply**, the endpoints of the line will snap to the points you entered, and the line will follow the mesh. Note that this option applies only to existing lines, so the setting is not saved when you hit the **Make default** button.

15.0 Surface plot attributes

15.1 Surface plot attributes menu item

This item pops up a window that allows you to set plot attributes for surface plots. If you select a surface plot from the **Active plots** list in the MeshTV **Main** window, then attributes you change and apply will be applied to the selected plot. Unselected surface plots will not be changed. If you select this item without selecting a surface plot first, you will have to select a surface plot before you can apply the changes. Note that if you make any changes and then select a surface plot, you will lose your changes since the window will update with the attributes of the selected plot.

You can modify the default settings for attributes of future plots by opening this window, making your changes, and then pressing the **Make default** button. This affects only future plots, not existing ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

15.2 Surface plot attributes window

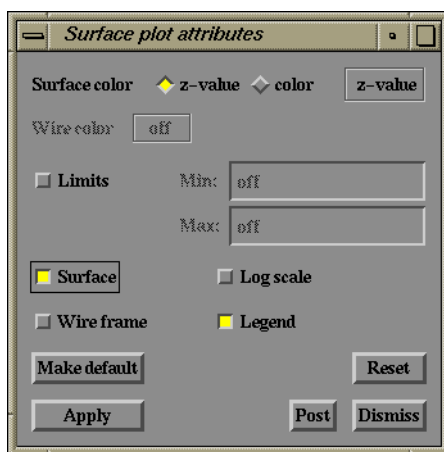


Figure 3-17: Surface plot attributes

The **Surface plot attributes** window looks like Figure 3-17, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing or future surface plots. The attributes include things like whether the legend will be shown, whether to show a wireframe on the surface, and what color to make the surface.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

15.3 Surf color button

This button allows you to select the surface color. This color can be either **Z value** or a solid color. If the desired color is **Z value**, click the "z-value" radio button. **Z value**, which is the default, colors the surface based on the value of the variable you have chosen for the surface plot. Note that this menu will be grayed out if the **Surface** toggle button is deselected. If a solid color is desired, click the "color" radio button and select a solid color by clicking the color button immediately to its right. Doing so will activate the **Palette editor** and a new surface color can be selected

15.4 Wire color button

This button allows you to select a new color for your wire frame using the **Palette editor**. Clicking the color button will activate the **Palette editor** so a new line color can be selected. Note that this menu will be grayed out if the **Wireframe** toggle button is deselected

15.5 Limits toggle

If you want to restrict the range of data used by the surface plot to generate its output, this is the option to use. Once you select the button, the **Min** and **Max** text fields become active, and you can enter in the range to use. You can change one or both fields.

15.6 Min text field

This text field becomes active once the **Limits** toggle button is selected. When the field contains the string "off," the actual data minimum is used, else the surface plot uses the number entered in the text field as the minimum number. When a user-supplied number is used, all data values below this minimum map to the same color as the user-supplied minimum. Note that setting a minimum also affects the vertical scaling of the plot.

15.7 Max text field

This text field becomes active once the **Limits** toggle button is selected. When the field contains the string "off," the actual data maximum is used, else the surface plot uses the number entered in the text field as the maximum number. When a user-supplied number is used, all data values above this maximum map to the same color as the user-supplied maximum. Note that setting a maximum also affects the vertical scaling of the plot.

15.8 Surface toggle

When this toggle is selected, the surface is drawn. You might want to turn this off if you wanted to see just the wire frame. You can modify the color of the surface via the **Surf color** options.

15.9 Wire frame toggle

When this toggle is selected, a wire frame of the surface is plotted. You can modify the color of the lines by using the **Wire color** button.

16.0 Vector plot attributes

16.1 Vector plot attributes menu item

This item pops up a window that allows you to set plot attributes for vector plots. If you select a vector plot from the **Active plots** list in the MeshTV **Main** window, then attributes you change and apply will be applied to the selected plot. Unselected vector plots will not be changed. If you select this item without selecting a vector plot first, you will have to select a vector plot before you can apply the changes. Note that if you make any changes and then select a vector plot, you will lose your changes since the window will update with the attributes of the selected plot.

You can modify the default settings for attributes of future plots by opening this window, making your changes, and then pressing the **Make default** button. This affects only future plots, not existing ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

16.2 Vector plot attributes window

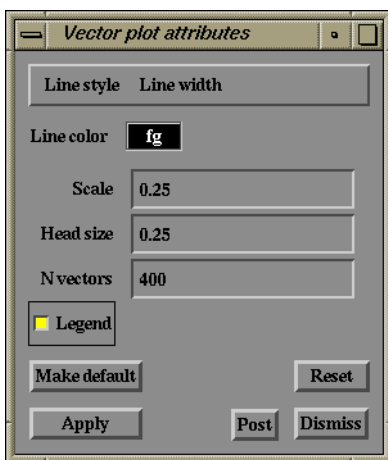


Figure 3-18: Vector plot attributes

The **Vector plot attributes** window looks like Figure 3-18, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing or future vector plots. The attributes include things like the scale (length) of the vectors, the head size of the vectors, and the total number of vectors to display.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

16.3 Scale text field

This text field governs the scale (length) of the vector.

16.4 Head size text field

This text field governs the size of the head on the vector. The default is 0.25, which means that the head takes up 1/4th of the length of the vector.

16.5 Nvectors text field

This text field allows you to select the number of vectors to display. The default is 400, though this number is too small for many plots. If you increase the number too much, however, the vectors will blanket the object, obscuring both themselves and any other plots.

17.0 Variables menu

17.1 Overview

The **Variables** menu presents alternate variables for a selected plot type, and the menu contains different variables depending upon which plot is selected. For example, if a Mesh plot is selected from the **Active plots** list, then the variables in this menu will be mesh variables from the open file. If a pseudocolor plot is chosen instead, then this menu will contain variables suitable for pseudocolor plots, like density or pressure. If you have selected a pseudocolor plot and click on a variable in the **Variables** menu, then the pseudocolor plot will be redrawn using the new variable. This allows you to switch to a new variable while keeping the same attributes for the plot.

Chapter 4

Operators and Operator Attributes

1.0 Overview

This chapter covers the items found under the **Operator** and **Operator attributes** menus in the **Active plots** area of the MeshTV **Main** window. The pulldown menus are shown in Figure 4-1.

This section gives a quick description of each button item. More detailed descriptions follow this section.

Operators

The **Operators** pulldown menu allows you to select different plot operators, like Reflect or Index select.

Operator attributes

The **Operator attributes** pulldown menu allows you to pop up windows which let you set attributes for each operator.

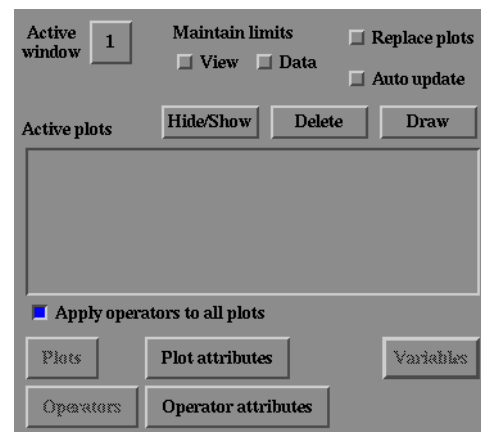


Figure 4-1: Active plots area

2.0 Operators menu

2.1 Overview

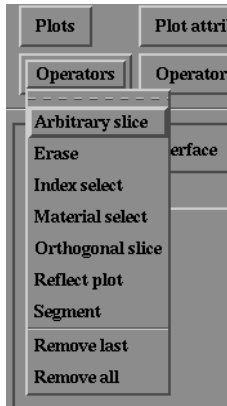


Figure 4-2: Operators

Selecting this menu item pops up a list of MeshTV operators. First highlight the plot to which you want to apply the operator, unless you have the **Apply operators to all plots toggle button** (located above the **Operators** menu beneath the **Active plots list**) turned on, in which case the operator will be applied to all plots. Select the operator you wish to apply. The plot will redraw with the operator applied. Note that this menu will be grayed out (and inaccessible) until you open a file.

The dotted line at the top of the menu denotes a "tear-off" menu, which means that when you click on the dotted line, the menu will "tear off" and stay posted until you close the window or press the ESC button while the window is selected.

Available operators are **Arbitrary slice**, **Erase**, **Index select**, **Material select**, **Orthogonal slice**, **Reflect plot**, and **Segment**. You can remove the last applied operator, or all the operators, but you cannot remove operators in an arbitrary order.

2.2 Arbitrary slice operator

This operator takes an arbitrary slice through a 3D data set. The coordinates of the slice plane are set in the **Arbitrary slice operator attributes** window. You can leave the slice in 3D or map it into 2D.

2.3 Erase operator

This operator allows you to remove a region of a 3D plot. The intersection of up to 6 regions defines the region to remove.

2.4 Index select operator

The index select operator performs a logical subsetting operation on a structured mesh. By specifying a minimum index, a maximum index, and an increment for each logical dimension of the mesh, you select the zones to display.

2.5 Material selection operator

The material selection operator removes materials from a mesh so they do not appear in a plot.

2.6 Orthogonal slice operator

This operator takes an orthogonal slice through a 3D data set. The axis along which the slice is taken (x, y, or z) is set in the **Orthogonal slice operator attributes** window.

2.7 Reflect plot operator

This operator reflects a plot across an axis.

2.8 Segment operator

This operator allows you to display a single zone and then to interactively add or subtract neighboring zones. It works only for 3D UCD datasets.

2.9 Remove last operator

When you select this option, the last applied operator is removed, and affected plots are redrawn.

2.10 Remove all operators

When you select this option, all operators are removed, and affected plots are redrawn.

3.0 Operator attributes menu

3.1 Overview

This pulldown menu provides access to windows which allow you to change operator attributes. Menu items with a ... in the name indicate that a window will open when the item is selected. The dotted line at the top of the menu denotes a "tear-off" menu, which means that when you click on the dotted line, the menu will "tear off" and stay posted until you close the window or press the ESC button while the window is selected.

The available options are **Arbitrary slice**, **Erase**, **Index select**, **Material selection**, **Orthogonal slice**, **Reflect plot**, and **Segment**. Each window allows you to set attributes for the operator type of the same name. The attributes can be applied to

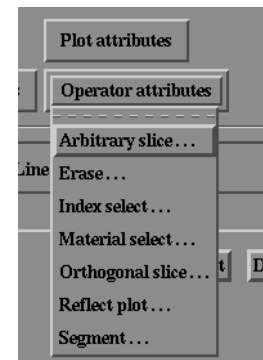


Figure 4-3: Operator attributes menu

existing operators, or they can become the default for operators assigned after changing the attributes.

4.0 Similar items

Many operators share similar items, like the **Apply** and **Dismiss** buttons. Rather than repeat the descriptions for these elements in the description for each window, common elements are described here, beginning with a table which details which windows include which elements, and ending with a description of the elements themselves.

MeshTV Plots	Apply	Dismiss	Make Default	Post	Reset
Arbitrary slice	•	•	•	•	•
Erase	•	•	•	•	•
Index Select	•	•	•	•	•
Material selection	•	•	•	•	
Orthogo- nal slice	•	•	•	•	•
Reflect plot	•	•	•	•	•
Segment	•	•	•	•	•

4.1 Apply button

Pressing the **Apply** button applies changes you made in the window containing the button. If you don't want your changes applied, press the **Dismiss** button without pressing **Apply**.

4.2 Dismiss button

When you wish to close a window, press the **Dismiss** button. You should press the **Apply** button before you press the **Dismiss** button if you want your changes applied. If you don't want to apply your changes, pressing only the **Dismiss** button (without pressing **Apply**) will return you to previously applied values.

4.3 Make default button

If you select this button, any changes you have made will become the default for all future operators of this type. Existing plots are unaffected, and these defaults last only for the duration of one MeshTV session. To permanently save the new defaults, you must select **Save settings** from the **File** menu after setting them with the **Make default** button.

4.4 Post button

If you plan to use a window repeatedly, but you don't want it cluttering up your screen, you might want to post it. When a window is posted, it appears in the **Notepad** area at the bottom of the MeshTV **Main** window, where it remains until it is unposted. Press this button to post the window. Once the window is posted, this button will change to **Unpost**.

4.5 Reset button

To restore a window to default settings, undoing any changes you have made, press the **Reset** button. To apply these default values to selected plots, press the **Apply** button. Default values are the ones in effect when you started MeshTV, unless the window has a **Make default button**, in which case the defaults will be the values in effect when you pressed this button, if you did.

5.0 Arbitrary slice operator attributes

5.1 Arbitrary slice operator attributes menu item

This item pops up a window which allows you to set operator attributes for arbitrary slice operators. If you select a plot that has an arbitrary slice operator applied from the **Active plots** list in the MeshTV **Main** window, then anything you change and apply will affect selected plots. Plots not highlighted will not be changed, unless **Apply operators to all plots toggle button** is on, in which case all plots will be changed. If you select this item without selecting a plot first, and you try to apply changes, a **Query window** will pop up asking if you want to add an arbitrary slice operator so you can apply the changes you have made. Note that if you make changes in this window and then select a plot that has an arbitrary slice operator applied, your changes will be lost.

You can modify the default settings for attributes for future operators by pulling up this window, making your changes, and pressing the **Make default** button. This affects only operators applied in the future, not already applied ones, and stays in effect only for the

remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

5.2 Arbitrary slice attributes window

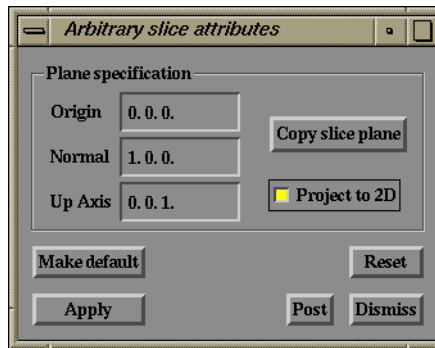


Figure 4-4: Arbitrary slice attributes

The **Arbitrary slice attributes** window looks like Figure 4-4, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing or future arbitrary slice operators. The attributes include things like the origin of the plane and the normal to it. You can also indicate whether you want to map the slice plane into 2D.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

5.3 Origin text field

This text field allows you to specify the origin of the plane. Generally, if you know the point through which you want the plane to pass, the coordinates for that point should be the coordinates for the origin.

5.4 Normal text field

This text field allows you to specify the normal to the plane. This orients the plane in space around the origin of the plane (not the origin of the problem space). This vector does not have to be a unit vector.

5.5 Up axis text field

This text field allows you to specify which direction is up. The up vector, projected onto the slice plane, points to the top of the screen when the plane is drawn in 2D. In fact, this field is only sensitive to input when the Map to 2D toggle is selected, since it only applies when you want to map the slice to 2D. You can change the slice's orientation, essentially rotating the plane, by choosing which direction will be up. The up vector does not need to be a unit vector, nor does it need to lie on the plane.

5.6 Map to 2D toggle button

This toggle button allows you to specify if the slice plane will be left in 3D, or if it will be projected into 2D. If you map the plane to 2D, you can use the Pick mode to query information from the slice. In particular, if you hold the middle mouse button down over the visualization window, a menu will pop up. The top choice in that menu will be **Mode**. If you click on that and pull over to the right, you'll see **Pick** as an option. Once you have selected that, your cursor will change to a crosshairs. You can position the crosshairs over the item in question and click the left mouse button on the slice. A marker will appear where you clicked. If you now click on the **Output window icon**, you can open the **Output window**, where information about the point you clicked will appear.

5.7 Copy slice plane button

This button allows you to apply the origin and normal from the last interactive arbitrary slice to your arbitrary slice attributes. This capability allows you to use the visualization window's interactive arbitrary slice mode to find the desired slice plane and then apply it to your plots for a higher degree of permanence.

6.0 Erase operator attributes

6.1 Erase operator attributes menu item

This item pops up a window which allows you to set operator attributes for erase operators. If you select a plot that has an erase operator applied from the **Active plots** list in the MeshTV **Main** window, then anything you change and apply will affect selected plots. Plots not highlighted will not be changed, unless **Apply operators to all plots toggle button** is on, in which case all plots will be changed. If you select this item without selecting a plot first, and you try to apply changes, a **Query window** will pop up asking if you want to add an erase operator so you can apply the changes you have made. Note that if you make changes in this window and then select a plot that has an erase operator applied, your changes will be lost.

You can modify the default settings for attributes for future operators by pulling up this window, making your changes, and pressing the **Make default** button. This affects only operators applied in the future, not already applied ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

6.2 Erase attributes window

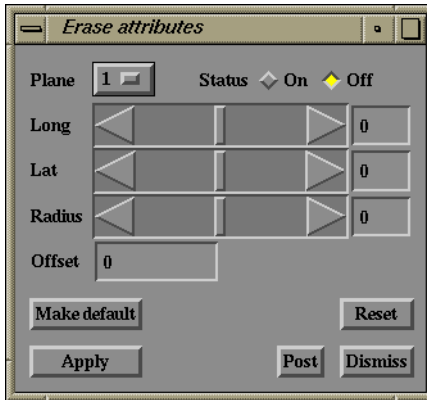


Figure 4-5: Erase attributes

The **Erase attributes** window looks like Figure 4-5, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing or future erase operators. The window includes controls to manage the intersection of up to 6 spaces.

The erase operator allows the erasure of a 3D region formed by intersecting up to 6 regions, each specified by the volume on one side of a plane. (This operator works only for 3D data files.)

A single plane eliminates the region on one side of a plane. Two planes eliminate a wedge shaped region. For example, using two planes that pass through the center of the viewing volume, oriented at 90-degree angles to each other eliminates, 1/4 of the viewing volume. Three planes eliminate a three-sided wedge region. Using three planes that pass through the center of the viewing volume, oriented at 90-degree angles to each other, eliminates 1/8 of the viewing volume.

The planes are positioned using the longitude, latitude, and radius settings.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

6.3 Plane option menu

Select the plane whose attributes you are setting by selecting the corresponding number from this option menu. The values range from 1 through 6.

6.4 Status radio buttons

This determines if a given plane is active. When you change the plane number via the **Plane option menu**, these radio buttons update with the status for that plane.

6.5 Longitude slider bar and text field

To position the erase plane, imagine that the object sits in space with the default 3D view where the positive Z axis points out of the screen, the positive X axis moves from the left to the right of the screen, and the positive Y axis moves from the bottom to the top of the screen.

The longitude control rotates the plane about the Y axis and can vary in value from -180 degrees to +180 degrees. Specifying a positive angle rotates the plane in a counter-clockwise fashion, when looking down on the Y axis.

You can either type your value into the text field, or use the slider to select a value.

6.6 Latitude slider bar and text field

To position the erase plane, imagine that the object sits in space with the default 3D view where the positive Z axis points out of the screen, the positive X axis moves from the left to the right of the screen, and the positive Y axis moves from the bottom to the top of the screen.

The latitude control rotates the plane about the X axis and can vary in value from -90 degrees to +90 degrees. Specifying a positive angle rotates the plane in a clockwise fashion, when looking down on the X axis.

You can either type your value into the text field, or use the slider to select a value.

6.7 Radius slider bar and text field

To position the erase plane, imagine that the object sits in space with the default 3D view where the positive Z axis points out of the screen, the positive X axis moves from the left to the right of the screen, and the positive Y axis moves from the bottom to the top of the screen.

The radius control moves the plane in and out along the direction specified by the longitude and latitude values. The radius can vary from -100 to +100. A value of 0 positions the plane at the center of the viewing volume. A positive radius moves the plane in the direction specified by the longitude and latitude values, while a negative radius moves the plane in the direction opposite the direction specified by the longitude and latitude values. A value of 100 corresponds to 1/2 the distance between any corner of the viewing volume and its opposite. This means the a value of 100 is guaranteed to be outside the volume and that a value between 0 and 100 may or may not be within the volume depending on the orientation of the plane.

You can either type your value into the text field, or use the slider to select a value.

6.8 Offset text field

When a non-zero offset is specified, the region specified by the planes is not removed from the problem, but is offset instead by the specified value in the direction derived by averaging the perpendiculars to each plane.

7.0 Index select operator attributes

7.1 Index select operator attributes menu item

This item pops up a window which allows you to set operator attributes for index select operators. If you select a plot that has an index select operator applied from the **Active plots** list in the MeshTV **Main** window, then anything you change and apply will affect selected plots. Plots not highlighted will not be changed, unless **Apply operators to all plots toggle button** is on, in which case all plots will be changed. If you select this item without selecting a plot first, and you try to apply changes, a **Query window** will pop up asking if you want to add an index select operator so you can apply the changes you have made. Note that if you make changes in this window and then select a plot that has an index select operator applied, your changes will be lost.

You can modify the default settings for attributes for future operators by pulling up this window, making your changes, and pressing the **Make default** button. This affects only operators applied in the future, not already applied ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

7.2 Index select attributes window

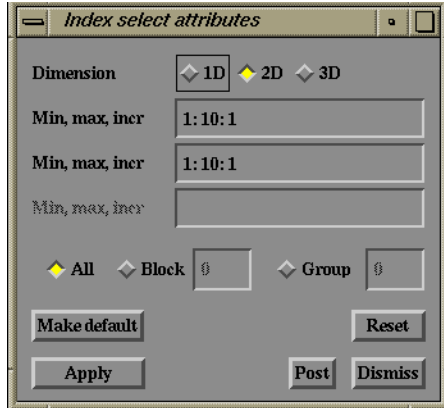


Figure 4-6: Index select attributes

The **Index select attributes** window looks like Figure 4-6, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing or future index select operators. The window includes controls to manage index selection for 1D, 2D, or 3D.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

7.3 Dimension radio buttons

The index select operator performs a logical subsetting operation on a structured mesh. These radio buttons allow you to indicate for how many dimensions you want to specify mesh subsetting.

7.4 Min, max, incr text fields

The index select operator allows you to specify a logical subsetting operation on a structured mesh by specifying a minimum index, a maximum index, and an increment for each logical dimension of the mesh. For example, if you have a 2D mesh, and you want only to show mesh lines for every other zone, you would specify the starting zone number, the ending zone number, and the number 2 in both text fields which are made active (not grayed out) when you select 2D in the **Dimension** radio buttons.

7.5 Hierarchy radio buttons

The **All**, **Block**, and **Group** radio buttons allow you to control index selection on multiblock meshes.

If your mesh is made of multiple blocks, each with independent logical indices, you can select an index range within a single block by selecting the **Block** radio button and entering the block number in the **Block** text field. In this case, the index values you specify are “local” to this block.

If your mesh is made of multiple groups, each (possibly) composed of multiple blocks with sequential logical indices within a single group, you can select an index range within a group by selecting the **Group** radio button and entering a group number in the **Group** text field. In this case, the index values you specify are “global” across blocks (although local to the group).

If you instead choose **All**, the same index range will be selected from every block or group, in the “global” indexing scheme, if possible. This is the value you should choose for single-block meshes.

8.0 Material selection operator attributes

8.1 Material selection operator attributes menu item

This item pops up a window which allows you to set operator attributes for material selection operators. If you select a plot that has a material selection operator applied from the **Active plots** list in the MeshTV **Main** window, then anything you change and apply will affect selected plots. Plots not highlighted will not be changed, unless **Apply operators to all plots toggle button** is on, in which case all plots will be changed. If you select this item without selecting a plot first, and you try to apply changes, a **Query window** will pop up asking if you want to add a material selection operator so you can apply the changes you have made. Note that if you make changes in this window and then select a plot that has a material selection operator applied, your changes will be lost.

8.2 Material selection attributes window

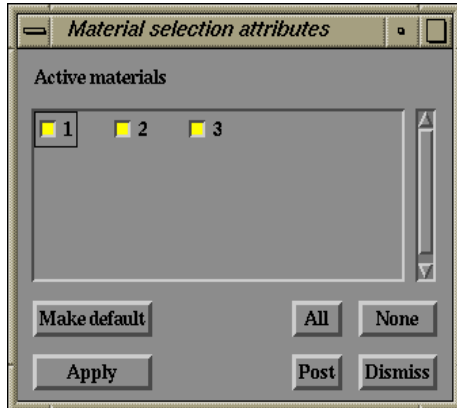


Figure 4-7: Material selection attributes

The **Material selection attributes** window looks like Figure 4-7. The list of the **Active materials** will contain entries if you have opened a file which has materials in it. If you are using a file with a different number of materials, the list of **Active materials** might be different.

This window allows you to set the active materials assigned to existing or future material selection operators.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

8.3 Active materials list

This scrolled list contains all of the materials in the problem. A toggle button appears next to each material number, and the state of the toggle indicates whether the material is active. When the toggle is “pushed in,” the material is active, and when it’s “popped out,” the material is inactive. Only active materials are plotted.

8.4 All button

Press this button to make all materials active, which is the default when MeshTV comes up. This is a quick way to make all your materials active, though you can also make materials active by “pushing in” the toggle button next to each material number.

8.5 None button

Press this button to make all materials inactive. You can use it when you want most materials to be inactive. First make them all inactive and then activate the ones you want by clicking on the toggle button next to them.

9.0 Orthogonal slice operator attributes

9.1 Orthogonal slice operator attributes menu item

This item pops up a window which allows you to set operator attributes for orthogonal slice operators. If you select a plot that has an orthogonal slice operator applied from the **Active plots** list in the MeshTV **Main** window, then anything you change and apply will affect selected plots. Plots not highlighted will not be changed, unless **Apply operators to all plots toggle button** is on, in which case all plots will be changed. If you select this item without selecting a plot first, and you try to apply changes, a **Query window** will pop up asking if you want to add an orthogonal slice operator so you can apply the changes you have made. Note that if you make changes in this window and then select a plot that has an orthogonal slice operator applied, your changes will be lost.

You can modify the default settings for attributes for future operators by pulling up this window, making your changes, and pressing the **Make default** button. This affects only operators applied in the future, not already applied ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

9.2 Orthogonal slice attributes window

The **Orthogonal slice attributes** window looks like Figure 4-8, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the attributes assigned to existing or future orthogonal slice operators. The window includes controls to define the axis along which to slice and the location of the slice plane via percent, zone number, or coordinate.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

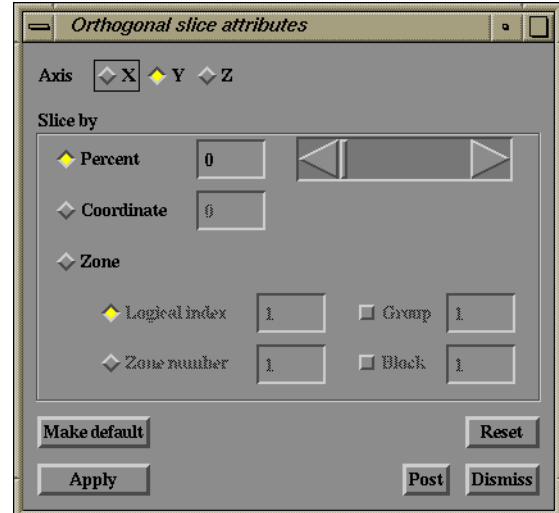


Figure 4-8: Orthogonal slice attributes

9.3 Axis radio buttons

The orthogonal slice operator slices through a 3D data set via a plane which is orthogonal to (perpendicular to) one of the coordinate axes. These radio buttons allow you to indicate which axis the plane will be orthogonal to.

9.4 Slice by label and radio buttons

The controls in this area allow you to specify how the slice plane moves along a chosen axis. You can select the placement of the plane by **Percent**, **Zone**, or **Coordinate**.

9.5 Slice by percent

When the **Percent** radio button is selected, you can specify the location of a slice plane along the chosen axis by percent along that axis. The slider allows you to select from the range 0 to 100, while the text field allows you to type in numbers within the same range.

9.6 Slice by coordinate

When the **Coordinate** radio button is selected, you can specify the location of a slice plane along the chosen axis by its coordinate value simply by typing the value in the text field.

9.7 Slice by zone

When the **Zone** radio button is selected, you can specify the location of a slice plane along the chosen axis by the index of a zone.

The **Logical index** radio button allows you to specify a zone in a structured mesh by its **i**, **j**, and **k** indices. If your data is also rectilinear, you need not specify all three indices; for example, if you have the **Y** axis selected you need only enter a **j** index. If your data has multiple groups, you must choose one of the groups by selecting the **Group** toggle button and entering a group number in the **Group** text field.

If your mesh is unstructured it cannot be accessed using logical indices. For this data type you must select the **Zone number** radio button.

The **Zone number** radio button allows you to specify a zone by its sequential index within a block. For example, on a curvilinear mesh with 10x10x10 zones, you can select the zone by its number in the range 0 to 999 (or 1 to 1000 in some datasets). If your data has multiple blocks, you must choose a block number by selecting the **Block** toggle button and entering a block number in the **Block** text field. Note, however, that on a rectilinear mesh this number is still a logical index along the selected axis, although it is now local to the selected block.

10.0 Reflect plot operator attributes

10.1 Reflect plot operator attributes menu item

This item pops up a window which allows you to set operator attributes for reflect operators. If you select a plot that has a reflect operator applied from the **Active plots** list in the MeshTV **Main** window, then anything you change and apply will affect selected plots. Plots not highlighted will not be changed, unless **Apply operators to all plots toggle button** is on, in which case all plots will be changed. If you select this item without selecting a plot first, and you try to apply changes, a **Query window** will pop up asking if you want to add a reflect plot operator so you can apply the changes you have made. Note that if you make changes in this window and then select a plot that has a reflect operator applied, your changes will be lost.

You can modify the default settings for attributes for future operators by pulling up this window, making your changes, and pressing the **Make default** button. This affects only operators applied in the future, not already applied ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

10.2 Reflect plot attributes window

The **Reflect plot attributes** window looks like Figure 4-9, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to specify where plots will appear. Clicking on a box, which highlights it, indicates that you want the plot to appear in that section.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

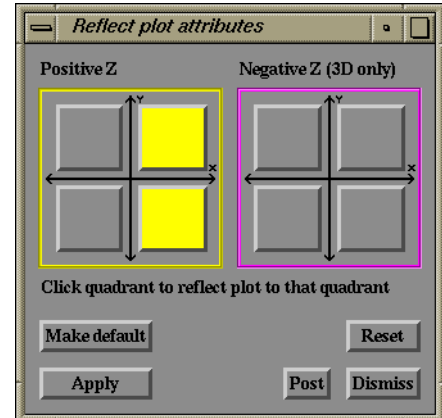


Figure 4-9: Reflect attributes

10.3 Positive Z

The top right quadrant of this section is the normal area to which MeshTV displays. For 2D problems, the **Positive Z** section is the only one that applies, whereas for 3D problems, the **Positive Z** section applies to the "front" of the screen. To reflect a plot into the lower right section, click the box to highlight it and then click **Apply button**. If you wish the plot to appear in just the lower right, then you must also click on the upper right box to remove the highlight. (And if you're doing 3D, you must make sure no boxes are high-

lighted in the **Negative Z** area.) Likewise, to reflect a plot into the upper left or lower left quadrants, you must click on those boxes and then press **Apply button**.

10.4 Negative Z

This section applies only to 3D plots, and it represents reflecting to the "back" of the screen. To reflect a plot into the back, lower right quadrant, click the lower right box to highlight it and then click **Apply button**. If you wish the plot to appear in just the back lower right, then you must make sure no other boxes are highlighted, including boxes in the **Positive Z** area. (If a box is already highlighted, just click on it again to remove the highlight.) Likewise, to reflect a plot into the upper left or lower left quadrants, you must click on those boxes and then press **Apply button**.

11.0 Segment operator attributes

11.1 Segment operator attributes menu item

This item pops up a window which allows you to set operator attributes for segment operators. If you select a plot that has a segment operator applied from the **Active plots** list in the MeshTV **Main** window, then anything you change and apply will affect selected plots. Plots not highlighted will not be changed, unless **Apply operators to all plots toggle button** is on, in which case all plots will be changed. If you select this item without selecting a plot first, and you try to apply changes, a **Query window** will pop up asking if you want to add a segment operator so you can apply the changes you have made. Note that if you make changes in this window and then select a plot that has a segment operator applied, your changes will be lost.

You can modify the default settings for attributes for future operators by pulling up this window, making your changes, and pressing the **Make default** button. This affects only operators applied in the future, not already applied ones, and stays in effect only for the remainder of the current MeshTV session unless you also select **Save settings** from the **File** menu.

Currently the segment operator only works for 3D UCD datasets.

11.2 Segment attributes window

The **Segment attributes** window looks like Figure 4-10, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to specify a starting zone from which to “grow” layers. For example, if you want to see all the neighboring zones to zone 20 in your problem, you specify zone 20 as your zone number, and set the layer slider to 1, indicated you want the search to go out one layer. Because of this layering capability, the operator is also called Onion Peel.

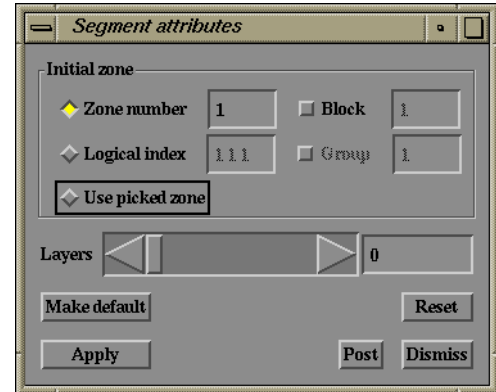


Figure 4-10: Segment attributes

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

11.3 Initial zone area

The controls in the area allow you to set the “seed” zone for the segment operator. The seed, or starting, zone sets the initial location to which layers can be added. When the segment operator is first applied, it displays only the initial starting zone, unless the user has modified the default settings for the operator. Note that if the mesh contains a large number of zones, the single zone MeshTV displays might take up only one pixel on the display, so you might think nothing has been displayed.

11.4 Zone text field

This text field allows you to set the seed zone for the segment operator. If you know the number of the zone you want to see, type it here. Note that if the mesh contains a large number of zones, the single zone MeshTV displays might take up only one pixel on the display, so you might think nothing has been displayed.

11.5 Block text field

If your data file contains blocks, you can set the toggle to indicate that you want to specify a block number. The **Block text field** allows you to type in that block number. The value in the **Zone text field** is then understood to be a zone in the supplied block.

11.6 Use picked zone toggle

You can set the seed zone for the segment operator by picking a zone in 3D pick. Once you have picked this zone, return the visualization window to **Navigate** mode, and then apply

the segment operator. Once you select this toggle, your picked zone will be the one displayed. Note that if the mesh contains a large number of zones, the single zone MeshTV displays might take up only one pixel on the display, so you might think nothing has been displayed.

11.7 Layers slider and text field

Use the **Layers** controls to display the zones that neighbor the starting zone. Each time you add a number to the number of layers, the segment operator will find and display all zones that are neighbors of the current set of zones displayed. A neighbor is any zone that shares a vertex. For example, if we're only displaying the seed zone, and we apply a layer of 1, the segment operator will find all zones that share a vertex with the seed zone. If we now increment the layer to 2, the segment operator will find all zones that share a vertex with any zone in layer 1.

Each collection of neighbors constitutes a new "layer" of zones. The number in the slider or text field indicates how many layers are displayed. If you decrease the number of layers, MeshTV "peels away" the outermost layer of zones.

Chapter 5

File Menu

1.0 Overview

This chapter covers items found under the **File** menu at the top of the MeshTV **Main** window. These items are: **Select file**, **Print window**, **Set print options**, **Save window**, **Set save options**, **Save movie**, **Save settings**, **Simulation**, and **Quit**. Menu items with a ‘...’ in the name indicate that a window will open when the item is selected. The dotted line at the top of the menu denotes a "tear-off" menu, which means that when you click on the dotted line, the menu will "tear off" and stay posted until you close the window or press the ESC button while the window is selected. The **File** menu window is shown in Figure 5-1.



Figure 5-1: File Menu

This section gives a quick description of each menu item. More detailed descriptions follow this section.

File menu

The **File** pulldown menu allows you to select the files you want to open, print results to a printer or to a file, save an image, save your current settings, access MeshTV's simulation capabilities, and exit the program.

Select file...

Selecting this menu item pops up a window which allows you to select the files you want to use during the current MeshTV session. Often, several files will already be selected when you start MeshTV. These files will be listed in the **Selected files** list located below the **File** pulldown menu in the MeshTV **Main** window.

Print window

Selecting this menu item prints the image from the active visualization window to the printer. The format of the image (e.g., PostScript, Raster PostScript) depends on what you specified via the **Set print options** window.

Set print options...

Selecting this menu item pops up a window which lets you set your print options, such as the name of the printer, whether to invert the background and foreground, and whether to do a screen capture.

Save window

Selecting this menu item saves the active visualization window. The image's attributes are governed by the options specified via the **Save image options** window. Depending on the file type chosen, either an image or the 3D graphics elements are saved.

Set save options...

Selecting this menu item pops up a window which lets you set your save options, such as the type of file to save, whether to invert the background and foreground, and whether to do a screen capture.

Save movie...

Selecting this menu item pops up a window which helps you generate an MPEG movie or a series of RGB or TIFF files.

Save settings

Selecting this menu item saves the current configuration for future use. For example, if you have the annotation window open and you select **Save settings**, then the annotation window will be open the next time you run MeshTV.

Simulation...

Selecting this menu item brings up a window which allows you to connect to any already running processes which interface with MeshTV, so you can view the results of a program's calculations interactively.

Quit

Exits MeshTV.

2.0 Select file

2.1 Overview

Selecting this menu item pops up a window which allows you to select the files you want to use during the current MeshTV session.

2.2 Select file window

This window is shown in Figure 5-2. The contents of the text fields and lists when you bring up MeshTV will probably differ from those in the picture.

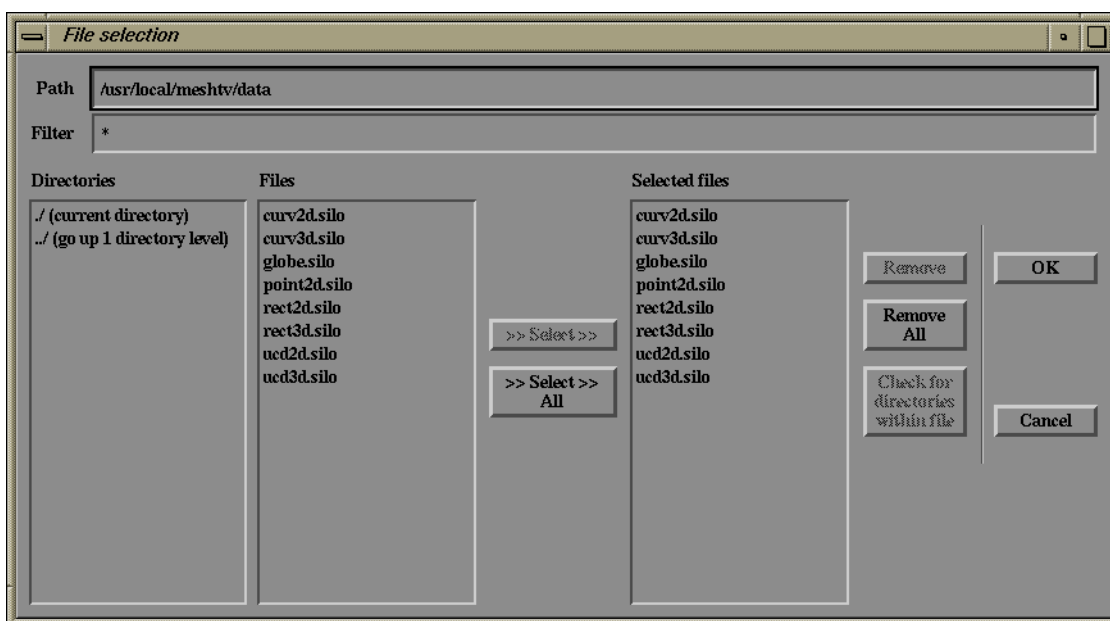


Figure 5-2: Select file window

The **Select file** window allows you to select files by browsing through your directory system. Often, several files will be pre-selected when you start MeshTV. These files will be listed in the **Selected files** list located below the **File** pulldown menu in the MeshTV **Main** window. They will also be listed in the **Selected files** list in this window.

2.3 Path text field

The **Path** text field allows you to enter a UNIX directory path name. By default, when you start MeshTV, the **Path** text field contains your current directory. To select files from a different directory, enter a path to the directory whose files you wish to use. You can use UNIX shell symbols, like the “~” for your home directory, or “../” to go up one from your current directory. Once you hit the enter key in this text field, the **Files** list updates with the files in that directory which match the requirements in the **Filter** text field. You can

have MeshTV use a specific directory by pulling up the **Preferences** window from the **Extra** menu, turning off the **Use current working directory toggle button**, and then selecting **Save settings** from the **File** pulldown menu in the MeshTV **Main** window. **Save settings** does not save the value in this text field unless you first turn off the **Use current working directory toggle button**.

2.4 Filter text field

The **Filter** text field allows you to determine which files in a directory will be placed into the **Files** list, which you will use to select files. The field accepts standard C-Shell UNIX pattern matching, where, for example, a “*” matches anything, and a “?” matches any single character. If your SILO files all contain a .silo extension, then a useful filter might be “*.silo”, which would match all files in that directory ending with .silo. **Save settings** will save the value in this text field at the time you select the **Save settings** menu item. The MeshTV default is “*”. You can specify more than one filter pattern.

2.5 Directories list

This lists UNIX directories in the directory specified in the **Path** text field. The “./” is the current directory, and the “../” is the parent directory. To see the contents of a given directory, either change the path in the **Path** text field and hit Enter, or double click on a directory in this list. You can also highlight (select) a directory and then hit the Enter key. The files in the directory you switch to will appear in the **Files** list.

2.6 Files list

This lists UNIX files in your current directory matching what is entered in the **Filter** text field. To see all files in the directory, remove any text from the **Filter** text field and hit the Enter key. There are two ways to select a file. The first is just to double click on the file you want. The second is a little more complicated. When you find a file you want to open, highlight (select) it by clicking once on the name. To select multiple files, either use Ctrl-click on each file (if they aren’t right next to each other), or select your first file and then Shift-click on another file to select all files in between the two. Once you have finished highlighting the files you want, press the **Select** button to move them into the **Selected files** list. If you want all the files, you do not need to highlight files first; just press the **Select all** button.

2.7 Select button

This button takes the files highlighted in the **Files** list and moves them to the **Selected files** list, which contains names of the files you want MeshTV to place in the MeshTV **Main** window. This button will be grayed out if you have yet to highlight any files.

2.8 Select all button

This button takes all files in the **Files** list and moves them to the **Selected files** list, which contains the files for MeshTV to place into the MeshTV **Main** window. The files in the **Files** list need not be selected first. This button will be grayed out if there are no files in the **Files** list.

2.9 Selected files list

This list contains the files which you have selected via the **Select** or **Select all** button, or by double-clicking on a file. These files will be placed into the MeshTV **Main** window **Selected files** list when the **OK** button is pressed. If you double click on a file in this list, MeshTV will check to see if there are any directories within the file, and if there are, the file will expand to the file name followed by a colon and the directory name.

2.10 Remove button

This button removes highlighted files from the **Selected files** list. This button will be grayed out if no files are selected.

2.11 Remove all button

This button removes all files from the **Selected files** list. Files in the **Selected files** list need not be selected to be removed. This button will be grayed out if there are no files in the **Selected files** list.

2.12 Check for directories within files button

Use this button to check if one or more selected files in the **Selected files** list contain SILO directories within them. If not, the files will be deselected and the button will grey out. If any files do contain SILO directories, those files will expand to the file name followed by a colon and the directory name. These entries can be individually removed if desired. The button will be grayed out if there are no highlighted files in the **Selected files** list.

2.13 OK button

Once you select your files, click on this button to approve your selections and dismiss the window. At this point, the files in the **Selected files** list will be placed into the MeshTV **Main** window **Selected files** list.

2.14 Cancel button

Press this button to cancel your actions. If you press this button, none of your selects will be used.

3.0 Print window

Selecting this item prints the image from the active visualization window to the printer specified via the **Set print options** window.

The active visualization window is the window listed beside the **Active window** label in the MeshTV **Main** window. Windows are numbered consecutively between 1 and 16, inclusive.

Note that if the **Screen capture toggle button** is on, you need to make sure that you have no windows obscuring your plots, since these will show up in the printed image. You can check the value of various print options by selecting the **Set print options...** menu item from the **File** menu.

4.0 Set print options

4.1 Overview

Selecting this menu item pops up a window that allows you to set various printer options, like the printer to which you want to print, the kind of output you want to produce, and whether or not to invert the foreground and background when you print.

4.2 Set print options window

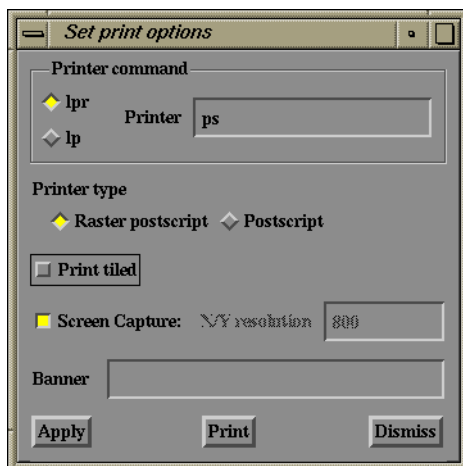


Figure 5-3: Set print options window

The **Set print options** window might look like Figure 5-3, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the options which govern the output when you print a plot. The options include things like the name of the printer, the type of output generated, and whether or not to print a banner at the top and bottom of the picture.

4.3 Printer command radio buttons

These radio buttons allow you to select UNIX commands to print files. If you are uncertain which one to use, check with your system administrator. If you are running MeshTV on a non-UNIX platform, save the image and then manually send it to the printer.

4.4 Printer text field

This text field takes the name of the printer to which you want to print. If you don't know what this is, you should see your system administrator for help. This name selects a specific printer on your network. That printer must also be able to handle the type specified by the **Printer type** option pulldown.

4.5 Printer type

These options determine which output type MeshTV will produce. Different printers require different output types, though some support many types. The available options are **Raster postscript** and **Postscript**.



Figure 5-4: Types

4.5.1 Raster postscript radio button

This option produces PostScript output in raster (pixel) form rather than the “normal” PostScript form, which details actual lines to draw. Because we are drawing pixels rather than lines, raster files can be smaller than conventional PostScript files in some circumstances, and therefore can take less time to print or move across a network. Raster PostScript files can be handled by PostScript printers.

When choosing between the two PostScript formats you should keep in mind the following trade-offs:

- PostScript files produce the highest quality images in 2D.
- Raster PostScript files are often large, take a long time to print, and can cause problems with most color PostScript printers when the image size is above 900 by 900. They can be smaller than PostScript files in certain situations, however.
- MeshTV automatically switches from PostScript to Raster Postscript when printing 3D images, since PostScript only gives wireframe images without hidden lines removed.

4.5.2 Postscript radio button

This option produces standard PostScript output, which can be handled by PostScript printers. Because this output type includes information as descriptions of lines to draw, files containing large amounts of information can become quite large, possibly even too large for a printer's buffer. If this occurs, you might want to try the raster PostScript type.

When choosing between the two PostScript formats you should keep in mind the following trade-offs:

- PostScript files produce the highest quality images in 2D.
- Raster PostScript files are often large, take a long time to print, and can cause problems with most color PostScript printers when the image size is above 900 by 900. They can be smaller than PostScript files in certain situations, however.
- MeshTV automatically switches from PostScript to Raster Postscript when printing 3D images, since PostScript only gives wireframe images without hidden lines removed.

4.6 Print tiled toggle button

This toggle button allows you to print all open visualization windows to a single sheet of paper.

4.7 Screen capture toggle button

This toggle button indicates whether the picture should be “grabbed” from the computer screen or taken from memory. If the picture is grabbed from the screen, the visualization windows will come to the front of the screen so that no other windows obscure them. Though grabbing the picture from memory yields a higher resolution image, grabbing via screen capture is faster, which is why it is the MeshTV default. If you turn off the screen capture, the image is rendered in software at 24 bits at the resolution specified by the **X/Y resolution text field**. Screen capture is supported for **Raster PostScript** only.

4.8 X/Y resolution text field

If you turn off screen capture via the **Screen capture toggle button**, the image is taken from memory at a specified resolution, which you can set by typing an integer number into this text field. Setting the X/Y resolution is supported for **Raster PostScript** only. Note that while higher resolutions produce nicer-looking pictures, they also increase file size.

4.9 Banner text field

Text entered into this text field will be printed at the top and the bottom of the image when it is printed.

4.10 Print button

Use this button to apply changes, print the image, and close the window.

4.11 Apply button

Pressing the **Apply** button applies changes you made in the window containing the button. If you don't want your changes applied, press the **Dismiss** button without pressing **Apply**.

4.12 Dismiss button

When you wish to close a window, press the **Dismiss** button. You should press the **Apply** button before you press the **Dismiss** button if you want your changes applied. If you don't want to apply your changes, pressing only the **Dismiss** button (without pressing **Apply**) will return you to previously applied values.

5.0 Save window

Selecting this menu item saves the active visualization window to a file. The saved format depends on what you specified via the **Set save options** window.

The active visualization window is the window listed beside the **Active window** label in the MeshTV **Main** window. Windows are numbered consecutively between 1 and 16, inclusive.

Note that if the **Screen capture toggle button** is on, you need to make sure that you have no windows obscuring your plots, since these will show up in the saved image. You can check the values of various save options by selecting the **Set save options...** menu item from the **File** menu.

6.0 Set save options

6.1 Overview

Selecting this item pops up a window which allows you to set various options, like the name of the file you want to save, the type of output to produce, and whether or not to invert the foreground and background when you save your image.

6.2 Set save options window

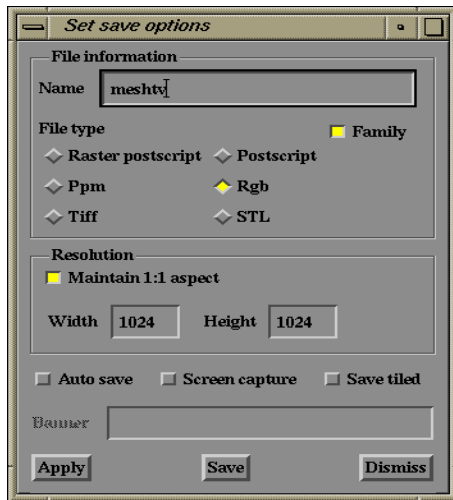


Figure 5-5: Set save options window

The **Set save options** window might look like Figure 5-5, if you are using MeshTV’s defaults. If you aren’t, the values might be different.

This window allows you to set the options in effect when you save a plot. The options include things like the name of the file to which to save, the type of output generated, and whether to print a banner at the top and bottom of the picture.

6.3 Name text field

This text field takes the name of the file you want to save. The name must be a valid UNIX file name. An extension appropriate to the file type will be added to the end of this name when the file is actually

saved. Raster Postscript and Postscript add a .ps extension, while Rgb adds an .rgb extension, Tiff adds a .tif extension, and STL adds an .stl extension.

6.4 File type

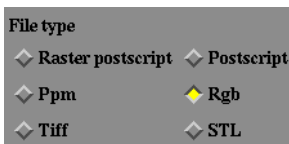


Figure 5-6: Types

These options determine the output type MeshTV produces. The available options are **Raster postscript**, **Postscript**, **Ppm**, **Rgb**, **Tiff**, and **STL**. (If you plan to print the file some time in the future, you should remember that different printers require different output types, so you should save your file in a format supported by the local printer you will use.)

6.4.1 Raster postscript radio button

This option produces PostScript output in raster (pixel) form rather than the “normal” PostScript form, which details actual lines to draw. Because we are drawing pixels rather than lines, raster files can be smaller than conventional PostScript files in some circumstances, and therefore can take less time to print or move across a network. Raster PostScript files can be handled by PostScript printers.

When choosing between the two PostScript formats you should keep in mind the following trade-offs:

- PostScript files produce the highest quality images in 2D.
- Raster PostScript files are often large, take a long time to print, and can cause problems with most color PostScript printers when the image size is above 900 by 900. They can be smaller than PostScript files in certain situations, however.

- MeshTV automatically switches from PostScript to Raster Postscript when printing 3D images, since PostScript only gives wireframe images without hidden lines removed.

6.4.2 Postscript radio button

This option produces standard PostScript output, which can be handled by PostScript printers. Because this output type includes information as descriptions of lines to draw, files containing large amounts of information can become quite large, possibly even too large for a printer's buffer. If this occurs, you might want to try the raster PostScript type.

When choosing between the two PostScript formats you should keep in mind the following trade-offs:

- PostScript files produce the highest quality images in 2D.
- Raster PostScript files are often large, take a long time to print, and can cause problems with most color PostScript printers when the image size is above 900 by 900. They can be smaller than PostScript files in certain situations, however.
- MeshTV automatically switches from PostScript to Raster Postscript when printing 3D images, since PostScript only gives wireframe images without hidden lines removed.

6.4.3 Ppm radio button

This option produces PPM (Portable Pixel Map) format files.

6.4.4 RGB radio button

This option produces SGI's RGB (Red, Green, Blue) format output, which is run-length encoded.

6.4.5 Tiff radio button

This option produces TIFF (Tagged Image File Format) files.

6.4.6 STL radio button

This option produces STL (Stereolithography) files. An STL file represents a 3D model, rather than a flat image like MeshTV's other file types (RGB, TIFF, etc.). Rapid prototyping machines use STL files to create physical parts.

The only MeshTV plots that can be saved to STL files are Mesh, Pseudocolor, and Filled boundary.

6.5 Family toggle button

This toggle button determines whether file families are generated. File families are sequentially numbered files, using the same base name. For example, if the base name in the **File** text field was “myfile” and the file type was **Postscript**, then the first time you saved, MeshTV would create a file called myfile0000.ps. The second time you saved a plot, the file would be called myfile0001.ps. This works well when you produce multiple, related plots, and you don’t want to write over a single file every time you save a plot.

6.6 Autosave toggle button

This toggle button indicates whether you want to automatically save plots without having to specifically issue a save command by selecting **Save image** in the **File** menu or **Save** from the **Set save options** window. Plots will be saved whenever you delete a plot from the visualization window or whenever you clear the visualization window completely. For example, if you added a pseudocolor plot and a mesh plot, an automatic save would occur if you deleted either one of the plots via the **Delete** button in the MeshTV **Main** window, or if you used the **Replace** button to replace either one of the two plots. An automatic save would also occur if you selected the **Clear** menu item from the MeshTV **Display** window

6.7 Save tiled toggle button

This toggle button allows you to save all open visualization windows to a single file which would fit onto a sheet of paper if printed.

6.8 Screen capture toggle button

This toggle button indicates whether the picture should be “grabbed” from the computer screen or taken from memory. If the picture is grabbed from the screen, the visualization windows will come to the front of the screen so that no other windows obscure them. Though grabbing the picture from memory yields a higher resolution image, grabbing via screen capture is faster, which is why it is the MeshTV default. If you turn off the screen capture, the image is rendered in software at 24 bits at the resolution specified by the Width and Height resolution text fields. Screen capture works for **Raster postscript** only.

6.9 Resolution

If you turn off screen capture via the **Screen capture toggle button**, the image is taken from memory at a specified resolution, which you can set by typing integer numbers into the width and height text fields. Note that when the **Maintain aspect toggle button** is turned on, the values typed for width and height will be forced to match. Setting the image width and height is supported for the **Raster postscript**, **RGB**, **TIFF**, and

PPM formats. Note that while higher resolutions produce nicer-looking pictures, they also increase file size.

6.10 Maintain aspect toggle button

This toggle button forces the **Resolution** text fields to match. Any value typed into the width text field will also be set into the height text field and vice-versa. To save images where the height does not equal the width, set this toggle to off before setting the image resolution.

6.11 Banner text field

Text entered in this text field will be printed at the top and the bottom of the image when it is saved. This can only be set if the file type is **Postscript** or **Raster postscript**.

6.12 Save button

Use this button to apply changes, save the image, and close the window.

6.13 Apply button

Pressing the **Apply** button applies changes you made in the window containing the button. If you don't want your changes applied, press the **Dismiss** button without pressing **Apply**.

6.14 Dismiss button

When you wish to close a window, press the **Dismiss** button. You should press the **Apply** button before you press the **Dismiss** button if you want your changes applied. If you don't want to apply your changes, pressing only the **Dismiss** button (without pressing **Apply**) will return you to previously applied values.

7.0 Save movie

7.1 Overview

Selecting this item pops up a window which allows you to set various options, like the name of the file you want to save, the type of output to produce, and whether or not to generate the movie right away or just to make a script which can be run later.

7.2 Save movie window

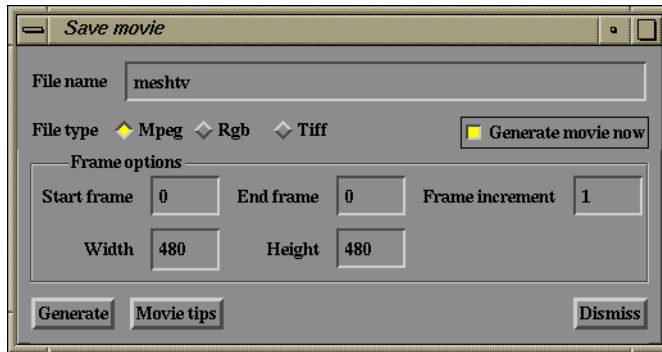


Figure 5-7: Save movie window

The **Save movie** window might look like Figure 5-7, if you are using MeshTV's defaults. If you aren't, the values might be different.

This window allows you to set the options in effect when you save an animation. The options include things like the name of the file to which to save, the type of output generated, and at what resolution to save the images.

7.3 File name text field

This text field takes the name of the file you want to save. The name must be a valid UNIX file name. An extension appropriate to the file type will be added to the end of this name when the movie file is generated. **Mpeg** adds the .mpeg extension and creates an mpeg movie. **Rgb** generates all the RGB files and names them with the file name followed by the frame number and a ".rgb" extension. **Tiff** generates all the TIFF files and names with the file name followed by the frame number and a ".tif" extension.

7.4 File type option

These toggle buttons allow you to choose the output type MeshTV produces. The available options are **Mpeg**, **Rgb** and **Tiff**.

7.5 Generate movie now toggle button

This toggle button determines whether the complete movie is generated when the **Generate** button is clicked, or whether MeshTV only creates files needed for the movie_generate script. The latter case, which happens when this toggle is off, lets the user call the movie_generate script at a later date. The user invokes movie_generate from the UNIX shell command line to generate the movie.

7.6 Frame options

These items allow you to select which files to use for your movie by providing both a starting and ending frame number, and also by allowing you to select an increment number. You can also set the resolution of the images which will be saved.

7.6.1 Start frame text field

This text field allows you specify which file in the family will become the starting frame of the movie. By default, it is the first file in the family, but you can start at any number.

This field takes an integer which represents the file's place in the list of files.

7.6.2 End frame text field

This text field allows you to specify which file in the family will become the ending frame of the movie. This number will be automatically set to the last file in the family, but if you want to end the movie before then, you can just change the number.

This field takes an integer which represents the file's place in the list of files.

7.6.3 Increment by text field

By default, MeshTV will create a frame for every file in the family of files. If you prefer, however, you can skip over some of the file, taking an image every two file, for example.

7.6.4 Resolution

The images for the movie are taken from memory at a specified resolution, which you set by typing integer numbers into the width and height text fields. Note that while higher resolutions produce nicer-looking pictures, they also increase file size.

7.7 Generate button

Use this button to generate the movie.

7.8 Movie tips button

Pressing this button pops up a window that contains helpful tips for generating movies.

7.9 Dismiss button

When you wish to close a window, press the **Dismiss** button. You should press the **Apply** button before you press the **Dismiss** button if you want your changes applied. If you don't want to apply your changes, pressing only the **Dismiss** button (without pressing **Apply**) will return you to previously applied values.

8.0 Movie tips

8.1 Movie tips window

The **Movie tips** window looks like Figure 5-8.

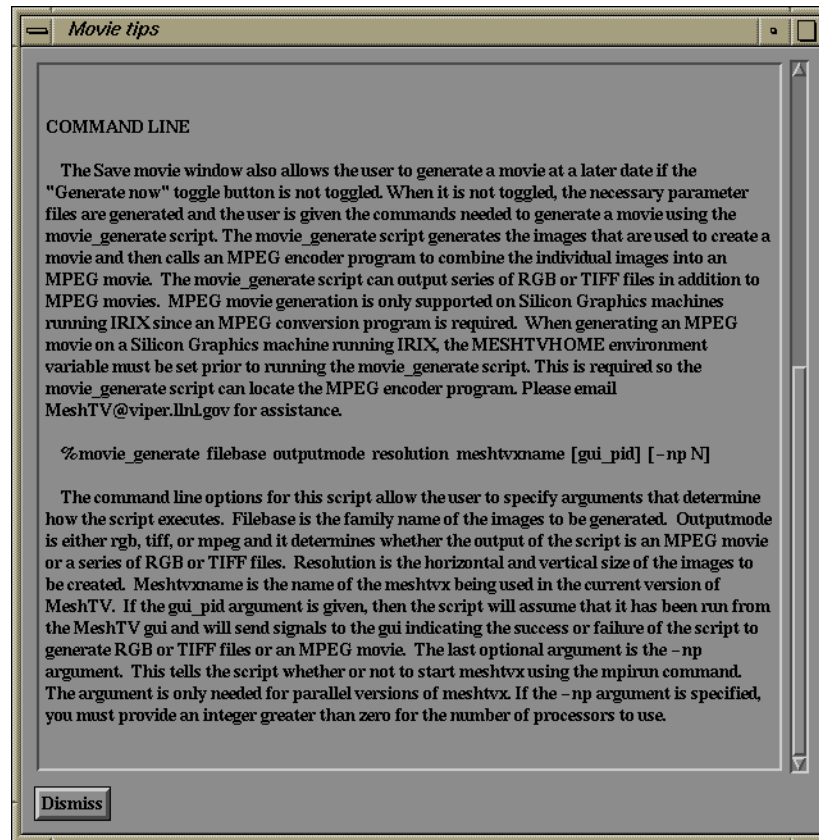


Figure 5-8: Movie tips window

This window provides help for generating movies.

9.0 Save settings

Selecting this item saves the current configuration for future use. For example, if you have the **Annotation** window open and you select **Save settings**, then the **Annotation** window will be open the next time you run MeshTV.

The things MeshTV saves when you save your settings include: the positions and dimensions of pop-up windows, whether the pop-up windows are visible or not, whether the pop-up windows are posted, and default settings for the attributes of all pop-up windows. MeshTV also saves items (e.g., perspective, full frame, bounding box navigation, and spin mode) from the menu that appears when you right click on the

visualization window. Things not saved include: the current visualization window, currently open files, currently active plots, and the number and locations of the output displays. An important thing to note is that the settings saved for the plot and operator attribute windows are the default settings, not the settings of the currently selected plot.

There are many different settings which MeshTV saves, so be aware that if you change the state of a radio buttons or text field and make the values the default by pressing the **Make default** button, these new values will probably be saved and used as the defaults the next time you run MeshTV. Some values are saved, like those in the **Preferences** window, and others are saved by virtue of having been applied— there is no **Make defaults** button in the window.

The “safest” way to use this option is to start MeshTV, change the settings you wish to save, and then select **Save settings**. This assures that you will only save the options you wanted to save.

10.0 Simulation

10.1 Overview

Selecting this menu item pops up a window which allows you to interact with simulations. A simulation is a running computer program which can send data to MeshTV.

10.2 Simulation window

The **Simulation** window might look like Figure 5-9.

The **Simulation** window allows you to manage how you interface with simulations (running computer programs that send data to MeshTV). It provides capabilities for pausing and continuing simulations.

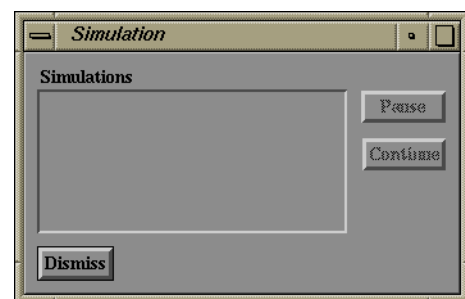


Figure 5-9: Simulation window

10.3 Simulations list

This contains a list of the running simulations you can manipulate. You must select a simulation before pressing either the **Pause** or **Continue** button, to indicate which simulation should pause or continue.

10.4 Pause button

Press this button to halt a running simulation. You must first click on the simulation from the **Simulations** list to select it.

10.5 Continue button

Press this button to continue a simulation which has been halted via the **Pause** button. This button only applies to a simulation selected from the **Simulations** list.

10.6 Dismiss button

When you wish to close a window, press the **Dismiss** button.

11.0 Quit

Select this menu item to quit MeshTV. MeshTV exits without prompting you to save or print anything. It also does not ask if you really want to quit, so make sure you're done before you select this.

Chapter 6

Controls Menu

1.0 Overview

This chapter covers the items found under the **Controls** menu at the top of the MeshTV **Main** window. These items are:

Animate, Annotation, Blocks, Color table, Expressions, Groups, Lighting, Line-out, Material, Palette editor, Powerwall, and Species. Menu items with a ... in the name indicate that a window will open when the item is selected. The dotted line at the top of the menu denotes a "tear-off" menu, which means that when you click on the dotted line, the menu will "tear off" and stay posted until you close the window or press the ESC button while the window is selected. The **Controls** menu window is shown in Figure 6-1.



Figure 6-1: Controls Menu

This section gives a quick description for each menu item. More detailed descriptions follow this section.

Controls menu

The **Controls** pulldown menu allows you to animate families of plots at different time steps, to annotate your plots, to work with blocks in a multiblock problem, to modify the color table used for pseudocolor plots, to create your own variables, and to manipulate materials and material species.

Animation...

Selecting this menu item pops up a window which allows you to set attributes for animating plots from files you have already opened.

Annotation...

Selecting this menu item opens a window which gives you control over MeshTV plot annotations.

Blocks...

Selecting this menu item pops up a window which allows you to manipulate blocks if your data file contains a multiblock problem.

Color table...

Selecting this menu item pops up a window which allows you to edit color tables or select a color table for pseudocolor and surface plots.

Expressions...

Selecting this menu item pops up a window which lets you define your own variables and perform operations on them or on existing variables.

Groups...

Selecting this menu item pops up a window which allows you to manipulate groups if your datafile contains meshes and variables in multiple groups.

Lighting...

Selecting this menu item pops up a window which allows you to manipulate light sources and shading of three-dimensional plots.

Line-out...

Selecting this menu item opens a window which lets you set globally-applying variables for reference lines and their accompanying curves.

Material...

Selecting this menu item opens a window which allows you to turn materials on and off.

Palette editor...

Selecting this menu item opens a window which allows you change the color palette for a visualization window or select colors for plot attributes.

Powerwall...

Selecting this menu item opens a window which allows you to manage how MeshTV windows are displayed on the Powerwall.

Species...

Selecting this menu item brings up a window which allows you to turn material species on and off.

2.0 Similar items

Many operators share similar items, like the **Apply** and **Dismiss** buttons. Rather than repeat the descriptions for these elements in the description for each window, common elements are described here, beginning with a table which details which windows include which elements, and ending with a description of the elements themselves.

MeshTV Plots	Apply	Dismiss	Post
Animation	•	•	•
Annotation	•	•	
Blocks		•	•
Color table	•	•	•
Expression	•	•	
Groups	•	•	•
Line-out	•	•	•
Materials	•	•	•
Palette editor	•	•	
Powerwall	•	•	
Species	•	•	•

2.1 Apply button

Pressing the **Apply** button applies changes you made in the window containing the button. If you don't want your changes applied, press the **Dismiss** button without pressing **Apply**.

2.2 Dismiss button

When you wish to close a window, press the **Dismiss** button. You should press the **Apply** button before you press the **Dismiss** button if you want your changes applied. If you don't want to apply your changes, pressing only the **Dismiss** button (without pressing **Apply**) will return you to previously applied values.

2.3 Post button

If you plan to use a window repeatedly, but you don't want it cluttering up your screen, you might want to post it. When a window is posted, it appears in the **Notepad** area at the bottom of the MeshTV **Main** window, where it remains until it is unposted. Press this button to post the window. Once the window is posted, this button will change to **Unpost**.

3.0 Animation

3.1 Overview

Selecting this menu item pops up a window which allows you to control the animation of plots.

3.2 Animation window

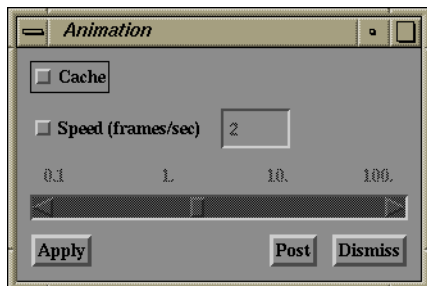


Figure 6-2: Animation window

The **Animation** window looks like Figure 6-2, if you are using MeshTV's defaults. If you aren't, the values might be different.

The **Animation** window allows you to indicate that plots should be cached and to control the speed of the animation by using a slider bar or typing a number.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

3.3 Cache toggle button

The **Cache** toggle button allows you to cache the polygons of your plots so animations run faster. After the original runthrough, when plots are first calculated and placed in the cache, the animation speeds up. To use this option, click on the toggle so that it is pressed "in" and becomes highlighted, generally in yellow. Note that this option can only be used on computers with sufficient RAM. This number is based on many things and can be

different on different machines, though a good “rule of thumb” is around 40 MEG of RAM. Your mileage may vary.

3.4 Speed Toggle button

Use this button to set your own frame rate for animations by moving the **Speed** slider bar or entering the number of frames/second in the **Speed** text field. If this button is off, the computer displays frames as fast as possible, and the **Speed** slider bar and **Speed** text field will be grayed out.

3.5 Speed text field

The **Speed** text field allows you to specify the number of frames to draw per second. If you specify a value higher than the computer can render, frames will be skipped. A value of 0 instructs the computer to go at its maximum speed without skipping frames, though you can also achieve this by deselecting the **Speed** toggle button. This field will be grayed out unless the **Speed** toggle button is selected.

The position of the slider bar is automatically updated to match the number typed in the text field when you hit the Enter key or **Apply** button.

3.6 Speed slider bar

The **Speed** slider bar allows you to specify the number of frames rendered per second. If you specify a value higher than the computer can render, frames will be skipped. A value of 0 instructs the computer to go at its maximum speed without skipping frames, though you can also achieve this by deselecting the **Speed** toggle button. This field will be grayed out unless the **Speed** toggle button is selected.

As you move the slider bar, the **Speed** text field will update with the currently selected number.

4.0 Annotation

4.1 Overview

Selecting this menu item pops up a window which allows you to control the annotation of plots.

4.2 Annotation window

The **Annotation** window looks like Figure 6-3, if you are using MeshTV's defaults. If you aren't, the values might be different.

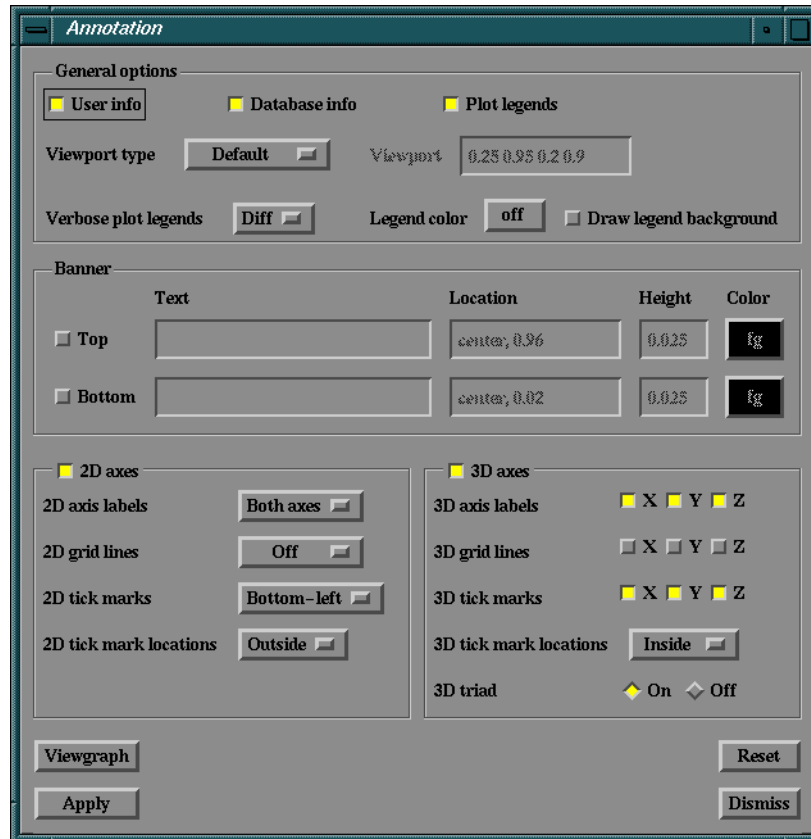


Figure 6-3: Annotation window

The **Annotation** window allows you to control general options, like whether to display file information, plot legends, and banners. It also allows you to control both 2D and 3D options, like whether grid lines and labels are shown, where tickmarks are placed if at all, and so forth. There is also a special button to set the attributes for making viewgraphs.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

4.3 General options

Items in this section apply to all plots, rather than being specific to 2D or 3D plots.

4.3.1 User info toggle button

This button turns user information legends on and off. The legend displays the name of the user and the current date.

4.3.2 Database info toggle button

This button turns database information legends on and off. This legend lists the filename, time, and cycle number of the problem, if they exist.

4.3.3 Plot legends toggle button

This button indicates whether or not to plot legends. This is a quick way to turn off all legends and then bring them back. To turn on/off individual legends, you need to bring up the associated plot attribute window. For example, for a particular boundary plot, you need to select that plot in the **Active plots** list and then bring up the **Boundary plot attributes**.

4.3.4 Viewport type options

There are three options for the viewport type. The standard option is **Default**, which is a viewport with coordinates of 0.2, 0.95, 0.15, and 0.9. A second option is **Full**, which makes the viewport as large as possible. This can lead to problems with legends, so you might have to turn those off. The last option is **User defined**, and when you select this option, the **Viewport** text field becomes sensitive. Viewports are specified by four numbers, each ranging from 0.0 to 1.0. The first and second of the four numbers are the x-min and x-max values, and the third and fourth numbers are the y-min and y-max values. The lower left-hand corner of the viewport is 0, 0, and the upper right-hand corner is 1, 1.

4.3.5 Viewport text field

This text field is only active when the **Viewport type** is set to **User defined**. Viewports are specified by four numbers, each ranging from 0.0 to 1.0. The first and second of the four numbers are the x-min and x-max values, and the third and fourth numbers are the y-min and y-max values. The lower left-hand corner of the viewport is 0, 0, and the upper right-hand corner is 1, 1.

4.3.6 Verbose label and option menu

This option controls when to print the database name, problem cycle, and problem time in individual plots. When this option is **On**, the information is always plotted for all appropriate plots. When this option is **Off**, the information is never plotted. When this option is set to **Diff**, MeshTV will only display the information for a plot when it differs from what is already displayed at the top of the graphical output window.

4.3.7 Legend background color buttons

These buttons control the legend's background color. When the legend is drawn with a background color, it is no longer transparent. This can make it easier to read the legend when the legend appears in front of a plot. By default, MeshTV does not draw a background behind legends.

I

4.4 Banner options

Items in this section apply to banners. Banners allow annotation text to be placed in the visualization window.

4.4.1 Banner display toggle button

This option controls when to plot banners in a visualization window. When this button is toggled, the banner for which it is set is displayed in the visualization window. When this button is not toggled, the banner is not displayed.

4.4.2 Banner text field

This option controls the text that is used for a banner. Up to fifty characters can be typed into this text field and used for the banner text.

4.4.3 Banner location text field

This option controls where a banner is positioned in a visualization window. The location is expressed as an x value followed by a y value. Both x and y values are floating point numbers between zero and one that correspond to the lower left corner of the banner. Instead of supplying a number for the x or y value, the word "center" can also be used. Using "center" causes the banner text to be centered in the dimension for which the word "center" was supplied.

4.4.4 Banner height text field

This option controls the height of a banner. The banner height is specified as a positive floating point number that represents a percentage of the visualization window height.

4.4.5 Banner color button

This option controls the banner color. Clicking on this button will bring up the **Palette editor** so a new banner color can be selected.

4.5 2D axes

This button provides a shortcut for turning 2D axes on or off. When the axes are off, the 2D axes options are grayed out.

4.5.1 2D axis labels options

This item allows you to select which axes in a 2D plot to label. You can label both axes by selecting **Both axes**, just the bottom axis by selecting **Bottom**, just the left axis by selecting **Left**, or you can turn labels off by selecting **Off**. The default is to label both axes.

4.5.2 2D grid lines options

This item allows you to select in which direction in a 2D plot to have grid lines. You can choose **X axis** to have grid lines just along the X axis, **Y axis** to have grid lines along just the Y axis, or **Both axes** to have grid lines along both axes. **Off**, which is the default, turns off all the grid lines.

4.5.3 2D tick marks options

With this option, you can place tick marks on axes in a 2D plot. You can choose **Off** to turn off tick marks, **Bottom** to have tick marks on just the bottom axis, **Left** to have tick marks on the left axis, **Bottom-left** to have tick marks on both axes, or **All** to have tick marks on all four axes. The default is **Bottom-left**.

4.5.4 2D tick mark locations options

This option allows you to specify where your tickmarks will be placed in a 2D plot. You may choose **Outside** to have tick marks outside the axes, **Inside** to have them inside the axes, and **Both** to have them appear both inside and outside the axes. This option is only active if at least one of the axes has tick marks.

4.6 3D axes

This button provides a shortcut for turning 3D axes on or off. When the axes are off, the 3D axes options are grayed out, except for the 3D triad option.

4.6.1 3D axis labels options

This item allows you to select which axes in a 3D plot to label. Click the box beside each axis you wish to label, or leave all unselected if you want no labels.

4.6.2 3D grid lines options

This item allows you to select the direction grid lines in a 3D plot. Click the box beside each axis you want to have grid lines, or leave them unselected if you want no grid lines.

4.6.3 3D tick marks options

With this option, you can specify which axes in a 3D plot will have gridmarks. Click the box beside each axis you want to have tick marks, or leave all the axes unselected if you want no tick marks.

4.6.4 3D tick mark locations options

This option allows you to specify where to place your tickmarks in a 3D plot. You may choose **Outside** to have tick marks outside the axes, **Inside** to have them inside the axes, and **Both** to have them appear both inside and outside the axes. This option is only active if at least one of the axes has tick marks.

4.6.5 3D triad radio buttons

This option allows you to turn the 3D triad on and off. The triad is an icon located in the lower left hand corner. It contains three lines which meet at one point and align with the X, Y, and Z axes. When you rotate a 3D image, the triad rotates and displays the orientation of the X, Y, and Z axes.

4.7 Viewgraph button

If you are printing viewgraphs, you might want to click this button. Once you click it, legends are turned off, grid tickmarks are added to the inside of all four axes, the viewport is set to a size that usually works well for viewgraphs, and the viewport type is set to full. If you don't want all of these options, you can always change them before you hit **Apply**.

4.8 Reset Button

To restore a window to default settings, undoing any changes you have made, press the **Reset** button. To apply these default values to selected plots, press the **Apply** button.

5.0 Block

5.1 Overview

Selecting this menu item pops up a window that allows you to choose the blocks you will use in your visualizations.

5.2 Block window

The **Block** window should look similar to Figure 6-4 when it pops up. The list of the **Active blocks** will contain an entry if you have opened a file with blocks.

This window allows you to select the blocks which will be active when plots are generated.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

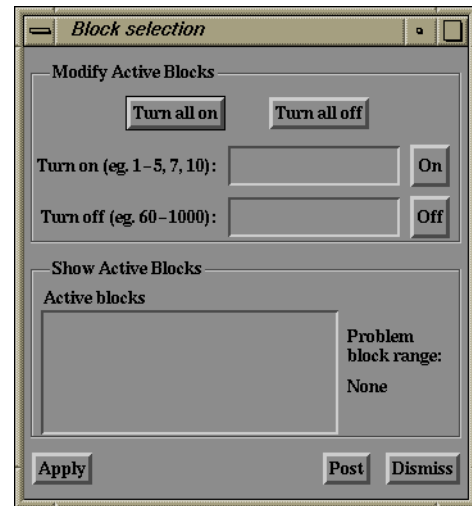


Figure 6-4: Block window

5.3 Turn all on button

Press this button to make all blocks active, the default for MeshTV. This is the quick way to make all your blocks active, though you can also make all blocks active by typing the full range into the **Turn on textfield**.

5.4 Turn all off button

Press this button to make all blocks inactive. You can use this when you want most blocks to be inactive. First make turn them all off and then activate the ones you want by typing the appropriate block numbers or ranges into the **Turn on textfield**.

5.5 Turn on textfield

Use this textfield to turn on blocks or ranges of blocks. For example, to turn on block 5, you can enter "5" and then hit enter or click on the **On button**. If you want to enter multiple blocks, like blocks 5, 7, and 26, you can type "5 7 26" (separated by spaces or commas) and hit enter or click on the **On button**. To turn on blocks 1 through 10, you would enter "1-10" or "1:10" and then hit enter or click the **On button**. Once finished, you'll see the appropriate entry appear in the **Active blocks list**. To make that selection active, you need to click on the **Apply button**.

5.6 On button

Use this button to turn on blocks or ranges of blocks. Once you enter a block or range of blocks into the **Turn on textfield**, click on this button to place the entry into the **Active blocks list**. Then, to make that selection active, you need to click on the **Apply button**.

5.7 Turn off textfield

Use this textfield to turn off blocks or ranges of blocks. For example, to turn off block 5, you can enter "5" and then hit enter or click on the **Off button**. If you want to turn off multiple blocks, like blocks 5, 7, and 26, you can type "5 7 26" (separated by spaces or commas) and hit enter or click on the **Off button**. To turn off blocks 1 through 10, you would enter "1-10" or "1:10" and then hit enter or click the **Off button**. Once finished, you'll see the appropriate blocks removed from the **Active blocks list**. To make that selection active, you need to click on the **Apply button**.

5.8 Off button

Use this button to turn off blocks or ranges of blocks. Once you enter a block or range of blocks into the **Turn off textfield**, click on this button to remove the blocks from the **Active blocks list**. Then, to make that selection active, you need to click on the **Apply button**.

5.9 Active blocks list

This scrolled list contains a list of all blocks you are currently plotting. When a file is first opened, that list contains the entire range of the problem. As you turn off some of the blocks, the list will update to show you which blocks are still active. For example, if your problem contains 36 blocks ranging from 0 to 35, you'll see an entry that reads "0-35". If you now turn off block 12, you'll see two entries. The first will read "0-11" and the second will read "13-35".

5.10 Problem block range label

This label shows you the block range for your current data file. For example, if your file contains 36 blocks ranging from 0 to 35, this text field will read "Block range: 0-35". When you don't have a file open, the range will read "Block range: None".

6.0 Color table

6.1 Overview

Selecting this menu item pops up a window that allows you to edit color tables or set a color table for pseudocolor and surface plots.

6.2 Color table window

The **Color table** window should look like Figure 6-5, if you are using MeshTV's defaults. If you aren't, the values might be different.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

This window allows you to select different color tables for pseudocolor and surface plots and edit new or existing color tables. You can even import color tables that conform to MeshTV's color table format. MeshTV supports color tables in the NCSA PAL binary format, which consists of 256 sets of 3 bytes. Each respective byte in the trio represents red, green, and blue for each of the 256 colors. MeshTV does not require the NCSA header at the beginning of the file.

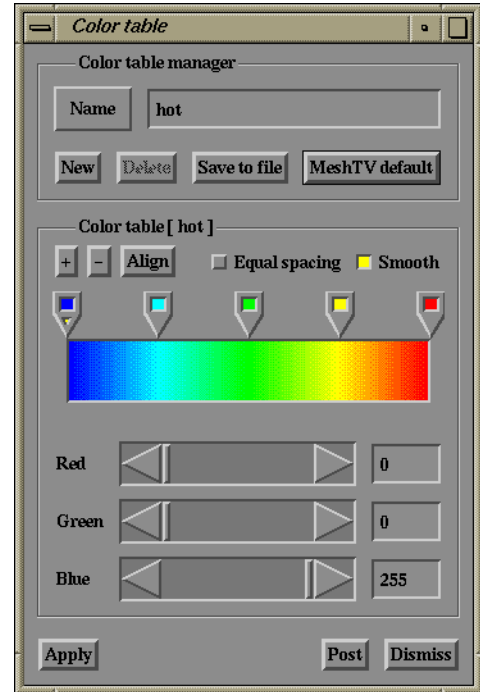


Figure 6-5: Color table window

6.3 Color table manager

This area consists of controls that let you select, create, delete, save, and reset color tables.

6.4 Color table selection option

The **Color table** selection option menu allows you to select different color tables for pseudocolor and surface plots. There are seven standard color tables and a user-defined menu that contains user-defined and external color tables, which allows you to select your own color table. When no color tables have been saved in the user menu, it is disabled until user-defined or external color tables are added.



Figure 6-6: Color tables

6.5 Cale black option

This color table mimics the color table used by the computer code, Cale, but the lowest level color is black. This allows low values to fade into the background when you're running with a black background in the visualization window.

6.6 Cale white option

This color table mimics the color table used by the computer code, Cale. This allows low values to fade into the background when you're running with a white background in the visualization window.

6.7 Contoured option

This color table provides four colors with a sharp transition to show course-grained changes in values.

6.8 Gray scale option

This color table ranges in color from black to white, in shades of gray. It can be useful to use this to check for features which might be difficult to see when using a color table with widely varying colors.

6.9 Hot to cold option

This color table ranges in color from blue to red, with blue being "cold," and red being "hot."

6.10 Rainbow option

This color table presents colors as they appear in a rainbow.

6.11 X-ray option

This color table ranges in values from white to black, in shades of gray. It can be used to simulate what a radiograph of the problem might look like.

6.12 User option

When you select this color table option, you are selecting a user-defined color table. The user-defined color table was either created with the color table editor or was read from an external file. If the color table was read from an external file, it must conform to MeshTV's color table format. MeshTV supports color tables in the NCSA PAL binary format, which consists of 256 sets of 3 bytes where each respective byte in the trio represents red, green, and blue for each of the 256 colors. MeshTV does not require the NCSA header at the beginning of the file.

6.13 File name text field

This text field allows you to enter a path to your own color table, enter the name of a new color table, or enter a filename to which the active color table will be saved. If you want to import your own color table from a file, type the path and name of the color table into this text field and press the Enter or Return key on your keyboard. Doing so will load the specified color table into the color table window and add it to the color table window's user-defined color table menu. If you want to enter the name of a new color table, simply type a new name into the text field and press the **New button**. If you want to save the currently active color table to a file, type a filename into the text field and press the **Save to file button**.

6.14 New button

This button allows you to create a new color table. To create a new color table, first type a new color table name into the **File name text field** and click the **New** button. If the color table name already exists, an information message will tell you to enter a different color table name. After a new name has been entered and the **New** button has been clicked, a copy of the currently active color table is made and given the new name. The new color table is added to the user-defined color table menu and can be modified as needed. The **New** button is disabled if the active color table was imported from a color table file since color table files do not contain the information necessary to edit a color table.

6.15 Delete button

This button allows you to delete a user-defined color table from the user-defined color table menu. When a user-defined color table is deleted, its name is removed from the user-defined color table menu and the active color table is set to "Hot to cold". If the currently active color table is one of the seven MeshTV standard color tables, the **Delete** button is disabled.

6.16 Save to file button

This button allows you to save the currently active color table to an external color table file. To save the currently active color table to an external color table file, type the desired filename in the **File name text field** and press the **Save to file button**.

6.17 MeshTV default button

This button restores a MeshTV standard color table to its default values. If the currently active color table is a user-defined color table, this button is disabled.

6.18 Color table editor

This area contains the controls necessary to edit a color table.

6.19 Color spectrum

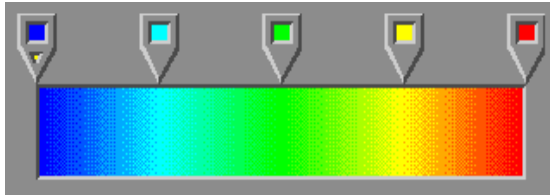


Figure 6-7: Color spectrum

All user-defined color tables are now defined in terms of a set of color control points. The color spectrum displays the color and location of these control points relative to one another. This allows the user to edit a color table by changing color control points. A color table always has at least two color control points, which are

shown in above figure as pointy boxes above the color spectrum. The color control points can only be edited one at a time and the color control point must be selected before it can be changed. The currently selected color control point is shown on the very left and has a small mark below its color box. To select a new color control point, simply click on one using the pointer.

Since the position of the color control points helps to determine the look of the color table displayed in the color spectrum, color control points can also be moved. To move a color control point, drag it to the desired location with the left pointer button. You can also click on an area where no color control point exists. This will move the selected control point closer to where the pointer was clicked.

Color tables that have been read from external color table files will not have color control points. This type of color table cannot be edited with the color spectrum.

6.20 Add color button

This button, drawn with a "+" character, inserts a new color control point into the color spectrum to the right of the selected color control point. The newly inserted color control point then becomes the selected color control point.

6.21 Remove color button

This button, drawn with a "-" character, removes the selected color control point. A different color control point is then selected.

6.22 Align button

This button replaces the position information of all of the color control points in order to make them equally spaced.

6.23 Equal spacing toggle button

This toggle button makes the color spectrum ignore the position information of the color control points and makes them equally spaced. When the button is not toggled, the position information is again taken into account.

6.24 Smooth toggle button

This toggle button turns color interpolation between control points on and off.

6.25 Color value scroll bar

The color value scroll bars indicate the color of the selected color control point. The color of the selected color control point can be changed by moving the scroll bars. There is a scroll bar for the red color channel, the green color channel, and the blue color channel.

6.26 Color value text field

The color text fields indicate the color of the selected control point as numbers from 0 to 255. The color of the selected color control point can be changed by typing new values into the color text fields. There is a text field for the red color channel, the green color channel, and the blue color channel

7.0 Expressions

7.1 Overview

Selecting this item brings up a window which allows you to create your own variables for MeshTV to plot.

7.2 Expression window

The **Expression** window should look like Figure 6-8, if you have never saved your own expressions via the **Save settings** menu item.

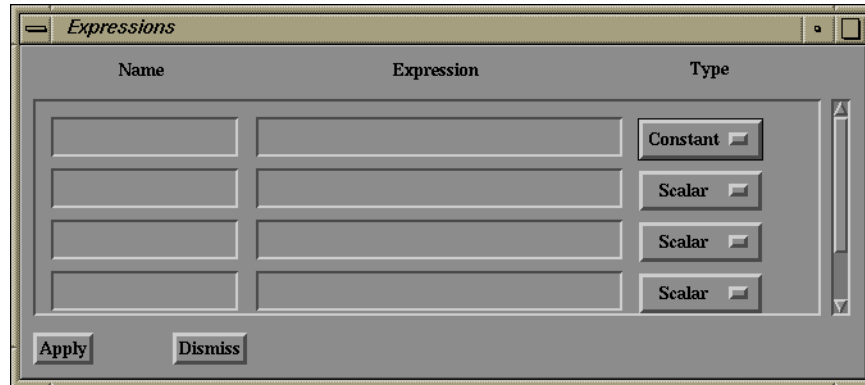


Figure 6-8: Expression window

The **Expression** window allows you to create your own variables from existing variables via mathematical and graphical operations. Once you create a variable, you can use it to create other variables.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

7.3 Name text field

This field contains the name of the variable you are creating. All variables must have their own, distinct name. Names can contain letters and numbers in any order, but there shouldn't be any "special" characters other than a dash or underscore.

7.4 Expression text field

This field contains the expression which creates the new variable. New variables are constructed from existing ones, either variables from the SILO file, or variables you previously created.

Expressions must be very specific. Because of this, they can become rather complicated. If you want to learn more about MeshTV expressions, see the *defvar* command in the *MeshTV Command Line Interface Manual*.

7.5 Type option

This option area indicates what type of variable is being created. There are five possibilities: **Constant**, **Material**, **Mesh**, **Scalar**, and **Vector**. You must specify the kind of variable you are creating so MeshTV will know how to generate and plot it.



Figure 6-9: Types

The **Constant**, **Material**, **Scalar**, and **Vector** data types in MeshTV refer to values placed at either the zones or the nodes of a mesh. To MeshTV, variables created with these data types are equivalent to quad/ucd/point variables and materials in a SILO data file.

Constant: The values at all nodes/zones are the same value.

Material: Each zone has a material number associated with it. You can use this existing material to create a new material.

Scalar: Each node/zone has one scalar value associated with it.

Vector: Each node/zone has a 2D or 3D vector associated with it.

The **Mesh** data type refers to meshes. To MeshTV, meshes created with this data type are equivalent to quadmesh and pointmesh variables in a SILO data file.

Mesh: Mesh nodes are laid out in a logically rectangular fashion.

8.0 Groups

8.1 Overview

Selecting this menu item pops up a window that allows you to choose the groups you will use in your visualizations.

8.2 Group window

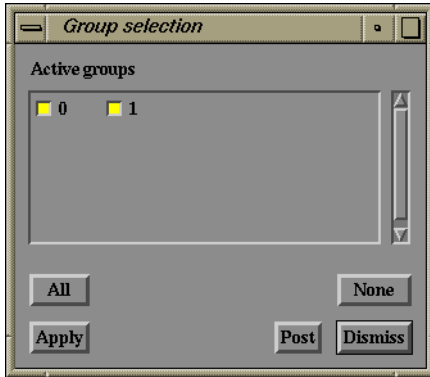


Figure 6-10: Group window

The **Group** window should look similar to Figure 6-10 when it pops up. The list of the **Active groups** will contain entries if you have opened a file with multiple groups.

This window allows you to select the groups which will be active when plots are generated.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

8.3 Active groups list

This scrolled list contains all of the groups in a problem. A toggle button appears next to each group number, and the state of the toggle indicates whether the group is active. When the toggle is “pushed in,” the group is active. When it’s “popped out,” the group is inactive. Only active groups are plotted.

8.4 All button

Press this button to make all groups active, the default for MeshTV. This is the quick way to make all your groups active, though you can also make groups active by “pushing in” the toggle button next to the group number.

8.5 None button

Press this button to make all groups inactive. You can use this when you want most groups to be inactive. First make them all inactive and then activate the ones you want by clicking on the toggle buttons next to them.

9.0 Lighting

9.1 Overview

Selecting this menu item pops up a window that allows you to manipulate the light sources and shading of three-dimensional plots.

9.2 Lighting window

The **Lighting** window should look similar to Figure 6-11 when it pops up. If you are using MeshTV's defaults, most controls in this window will be grayed out and disabled.

This window allows you to modify light source settings and attributes, which change the shading on many three-dimensional plots.

For every window, you can choose between two pre set lighting layouts or specify a custom one. If you decide to create your own, you have control over the type, direction, and brightness of up to eight different light sources. In addition, there is a preview area which indicates what the layout will look like when it is applied.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

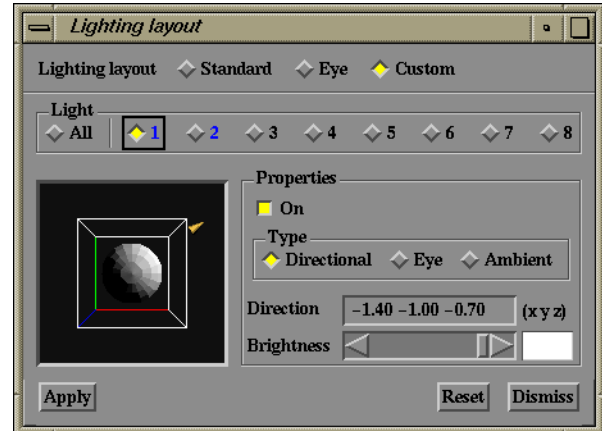


Figure 6-11: Lighting window

9.3 Lighting layout

There are two preset lighting layouts you can choose. The first is MeshTV's default lighting, labeled **Standard**. This layout has two opposite-facing white light sources that are positioned relative to the mesh. One points in the direction $\langle -1.4, -1.0, -0.7 \rangle$, and the other faces $\langle 1.4, 1.0, 0.7 \rangle$.

The second preset layout is entitled **Eye**. This layout has a single light source that is positioned relative to the viewer and that always points directly away from the viewer. This light source faces in the direction $\langle 0, 0, -1 \rangle$.

You can also choose to create a custom lighting layout for each window. If you select the **Custom** layout, then the remaining controls on the window become available to modify and preview light sources. You can create a different custom layout for each visualization window, and these will all be saved in your MeshTV configuration file.

9.4 Light selection

This control selects which light sources are shown in the preview area and are being modified. In addition, the labels for any light sources which are currently turned on will be in a different color.

When using a preset layout, this control is always set to show **All** light sources. When defining a custom layout, any option may be selected.

When this is set to **All**, all light sources will be shown in the preview area, and the controls to modify the light source properties will be disabled. When this is set to a single light source, **1** through **8**, then only that light will be shown in the preview area, and it can be modified through the **Properties** controls.

9.5 Light properties

When a custom light source has been selected, the controls in this area of the window can be used to modify the properties of that light source.

The toggle button labeled **On** turns on and off the light. Once a light has been turned on, it will be visible in the preview window and you can modify other properties of that light.

The light **Type** can be **Directional**, **Eye** or **Ambient**. A **Directional** light is one that stays fixed to the model. An **Eye** light is one which stays fixed to the viewer. An **Ambient** light has no direction; it is used to set a background level of illumination so that unlit areas are not completely black.

When the light is a directional or eye light, the **Direction** can be modified. You can enter the *x*, *y*, and *z* components of its direction in this text field. If the light is an ambient light, it has no direction and this field will be disabled.

The **Brightness** value determines the intensity of the illumination from this light. A very low value (towards the left) adds almost no illumination to the scene, and a high value adds the most. The indicator to the right of this slider will show black for a low brightness and white for a high brightness.

9.6 Light preview

This area shows what the cumulative effect of the currently selected lights will have on the scene. It also shows the types and positions (when applicable) of all selected lights.

Several objects are drawn in this preview area. The sphere is illuminated with the active light sources to give a rough impression of the shading effect on the scene. It is surrounded by a box which helps give orientation information. The *x*-axis is colored red, the *y*-axis green, and the *z*-axis blue. Directional lights are shown as yellow cones, and eye lights are shown as blue arrows. Any ambient lights are shown as light bulbs in the lower left corner of this preview area.

When **All** lights are selected, this area will show all lights which are turned on. In this mode, when the *left* mouse button is pressed and held in this area, dragging the mouse will

rotate the scene so that you may see it from any angle. When the *middle* mouse button is pressed in this area, the rotation will return to the normal angle.

When a single light, **1** through **8**, is selected, this area will show the selected light if it is on. In this mode, when the *left* mouse button is pressed and held in this area, dragging the mouse will change this light source's direction. (If the active light is an ambient light, this has no effect.) The *middle* mouse button will reset the rotation to the normal angle when pressed.

10.0 Line-out

10.1 Overview

Selecting this item pops up a window which allows you to control the mode of operation for line-outs, which are also known as reference lines and curves.

10.2 Line-out window

The **Line-out** window looks like Figure 6-12, if you are using MeshTV's defaults. If you aren't, the values might be different.

The **Line-out** window allows you to indicate whether reference line plots and curve plots should be replaced when regenerated, whether the reference line is specified by a straight line or by following the (logical) mesh, and the output window for curve plots that are generated from the reference line. There is also a toggle which allows you to direct MeshTV to use the first unused window, and if such a window doesn't exist, to open a new one.

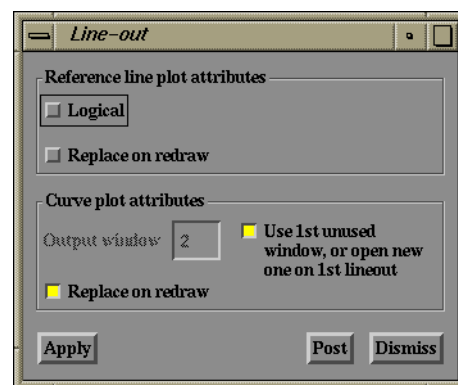


Figure 6-12: Line-out window

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

10.3 Reference line plot attributes area

This area contains options which apply to future reference line plots. Reference lines are the lines that appear in the visualization window when you switched the window to line-out mode and draw a line.

10.3.1 Logical toggle button

When you switch a visualization window to line-out mode (holding down the right mouse button and selecting **Mode** and **Line-out** from the pop-up menu), you can draw lines in that window. By default, these line will draw straight. They start at the endpoint created when you clicked the left mouse button, and follow the movement of your mouse as you hold the button down. The endpoint is indicated when you release the mouse button. However, if you select **Logical** and then draw your line, once you have finished drawing, the line will snap to the “logical” line so that it follows the mesh.

10.3.2 Replace on replot toggle button

When you switch a visualization window to line-out mode (holding down the right mouse button and selecting **Mode** and **Line-out** from the pop-up menu), you can draw lines in that window. By default, each time you start the process, you’ll end up with a new line. If you select **Replace on replot**, then the last reference line drawn will be replotted when you draw a new one.

10.4 Curve plot attributes area

This area contains options which apply to future curve plots. Curve plots are generated from reference lines. Reference lines are the lines that appear in the visualization window when you switch the window to line-out mode and draw a line.

10.4.1 Output window label and text field

When you switch a visualization window to line-out mode (holding down the right mouse button and selecting **Mode** and **Line-out** from the pop-up menu), you can draw lines in that window. Corresponding curves are drawn in the visualization window specified by this text field. Note that this window id number must differ from the number of the window in which you’re drawing your reference line, and it must also be the number of a window that either currently exists, or that is going to exist before you draw your first reference line. The Output window label and text field are unused when the **Use 1st window toggle button** is on, so in that case, they are "greyed out."

10.4.2 Use 1st window toggle button

When you switch a visualization window to line-out mode (holding down the right mouse button and selecting **Mode** and **Line-out** from the pop-up menu), you can draw lines in that window. Corresponding curves must be drawn to a different window from the window in which you’re drawing your reference line. When this toggle is selected, MeshTV places curves into the first unused (empty) window, when such a window exists. If no empty windows exist, then MeshTV opens a new window when the line-out is done, and this becomes the window into which curves are drawn. The Output window label and text field are "greyed out" when this toggle is on.

10.4.3 Replace on replot toggle button

When you switch a visualization window to line-out mode (holding down the right mouse button and selecting **Mode** and **Line-out** from the pop-up menu), you can draw lines in that window. Corresponding curves are drawn in the visualization window specified by the **Output window** text field. By default, each time you draw a reference line, you'll generate a new curve. If you select **Replace on replot**, then the last curve drawn will be replotted when a new one is generated.

11.0 Materials

11.1 Overview

Selecting this menu item opens a window which allows you change the active materials for the material selection operator.

*For the rest of the window description, see **Material selection operator attributes** in chapter 4.*

12.0 Palette editor

12.1 Overview

Selecting this menu item opens a window which allows you change the color palette for a visualization window or select colors for plot attributes.

12.2 Palette editor window

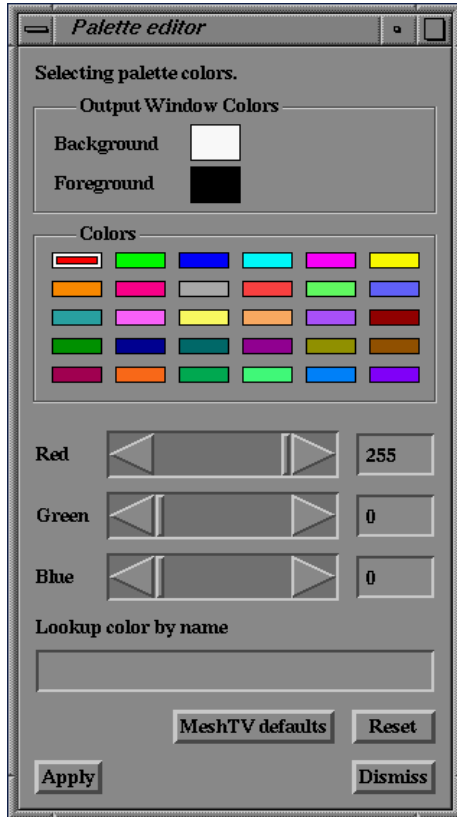


Figure 6-13: Palette editor window

The **Palette editor** window should look similar to Figure 6-13 when it pops up. The colors may be different if you have saved a custom palette.

The **Palette editor** window allows you change the color palette for a visualization window or select colors for plot attributes. Note that the Palette editor window has hints associated with it. If hints are disabled, they can be enabled in the **Preferences** window. Hints are displayed by moving the pointer over an area of interest

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

12.3 Select mode label

The select mode label indicates what function the Palette editor is performing. If the label says "Selecting palette colors" then the Palette editor is not associated with any plot attribute window and colors can be modified without affecting color attributes in other windows. If the label says

something else, the Palette editor is selecting a color for another window. If the selected color is changed and **AutoUpdate** is on, the change will be applied to the window for which the Palette editor is selecting a color. The select mode can be restored to "Selecting palette colors" by dismissing the Palette editor window with the **Dismiss** button and bringing the window up again with the **Palette editor** button in the **Controls Menu**.

12.4 Output window colors

This area of the Palette editor contains the colors that are used for the background and foreground colors of the active visualization window.

12.5 Background color

The background color corresponds to the background color of the active visualization window. To change the background color, first select it by clicking on the small color rectangle next to the "Background" label. When it is selected it will have a small white rectangle around it. Next, move the **Color value** sliders or enter new RGB color values

into the **Color text** fields. After changing the color, click the **Apply** button to set the color, or press the **Reset** button to revert to the last applied color palette.

12.6 Foreground color

The foreground color corresponds to the foreground color of the active visualization window. To change the foreground color, first select it by clicking on the small color rectangle next to the "Foreground" label. When it is selected it will have a small white rectangle around it. Next, move the **Color value** sliders or enter new RGB color values into the **Color text** fields. After changing the color, click the **Apply** button to set the color, or press the **Reset** button to revert to the last applied color palette.

12.7 Colors

This area contains the color palette for the active visualization window. The colors in this area are numbered 1 to 30. Color 1 is in the upper left corner and color 2 is to its right, and so on. To change a color, first select it by clicking it. When it is selected it will have a small white rectangle around it. Next, move the **Color value** sliders or enter new RGB color values into the **Color text** fields. After changing the color, click the **Apply** button to set the color, or press the **Reset** button to revert to the last applied color palette.

If **AutoUpdate** is on and the Palette editor's **Select mode** is not "Selecting palette colors", selecting a color in this area will update the window for which the Palette editor is selecting a color. If **AutoUpdate** is not on, then the **Apply** button must be clicked to update the other window.

12.8 Color value

The color value sliders are scroll bars that change the red, green, blue components of the selected color. Changing the positions of the sliders updates the values in the **Color text** fields and changes the color that is used to display the selected color. The color will not be saved into the applied palette until the **Apply** button is clicked. The **Reset** button changes the palette back to the last applied palette and undoes any changes that were done by changing the color value sliders.

12.9 Color text

The color text fields are text fields that change the red, green, blue components of the selected color. Changing the values in the text fields updates the positions of the **Color value** sliders and changes the color that is used to display the selected color. Red, green, blue values must be entered as integers from 0 to 255. Values outside of that range will not update the selected color. The color will not be saved into the applied palette until the **Apply** button is clicked. The **Reset** button changes the palette back to the last applied palette and undoes any changes that were done by changing the color text fields.

12.10 Lookup color by name

The lookup color text field is a text field that changes the red, green, blue components of the selected color by searching for a named color and retrieving its red, green, blue components. Entering a valid X color name (e.g. PeachPuff) into the text field causes a color lookup. If the lookup is successful, the selected color's red, green, blue components are updated and the text field is blanked. If the color lookup fails, then an error message is displayed and the color name can be edited. A list of X color names is supplied with MeshTV in a file called rgb.txt.

12.11 MeshTV default button

The MeshTV defaults button is a button that returns all of the colors in the active palette to the default MeshTV colors. This was added to provide an easy way to return to the default colors if a large number of them have been modified.

13.0 Powerwall

13.1 Overview

Selecting this menu item pops up a window which allows you to manage how MeshTV windows are displayed on the Powerwall.

13.2 Powerwall window

The **Powerwall** window should look similar to Figure 6-14 when it pops up. The layout and mapping may be different if you have saved custom settings.

The **Powerwall** window allows you to manage how MeshTV windows are displayed on the Powerwall. Note that the Powerwall window has hints associated with it. If hints are disabled, they can be enabled in the **Preferences** window. Hints are displayed by moving the pointer over an area of interest.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

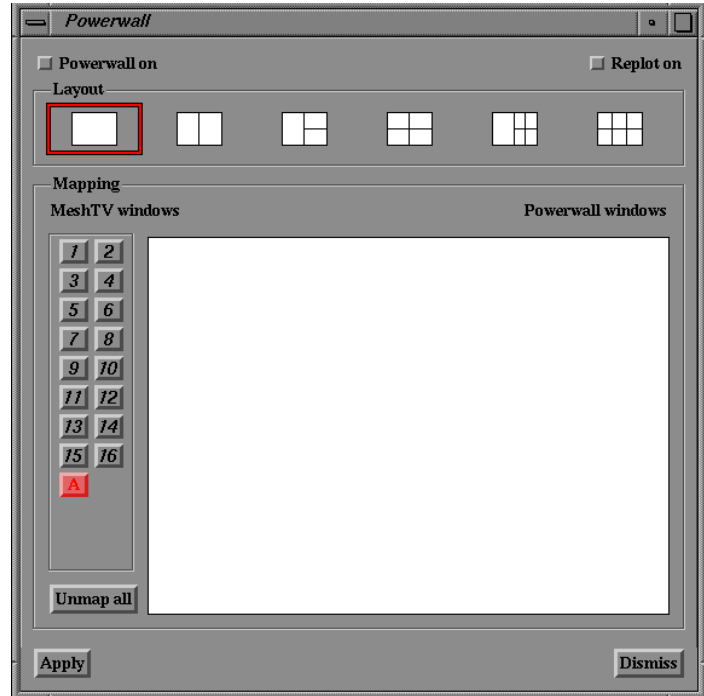


Figure 6-14: Powerwall window

13.3 Powerwall on

Toggle this button to turn the Powerwall on or off. If you are running on a machine that is not capable of using the Powerwall, then toggling this button has no effect.

13.4 Powerwall replot

Toggle this button to ensure that plots are redisplayed when the Powerwall layout changes. If this button is toggled on, the Powerwall will redisplay plots when the layout changes. This ignores MeshTV window to Powerwall canvas mapping. If this button is not toggled, only the active MeshTV window will be displayed when the Powerwall layout changes.

13.5 Layout area

The Layout area is depicted in Figure 6-15.



Figure 6-15: Powerwall layouts

The layout area contains pictures of different Powerwall layouts. Clicking on a layout picture selects a Powerwall layout and updates the Powerwall window.

13.6 Mapping

The mapping area of the Powerwall window contains features needed to map MeshTV windows to Powerwall canvases.

13.7 Token

Tokens are shown in Figure 6-16.



Figure 6-16: Tokens

A token is a small label that represents either a MeshTV window or the Powerwall active canvas. A Powerwall canvas is a region on the actual Powerwall. Use the middle mouse button to drag a token to an appropriate drop site. Tokens numbered from 1 to 16 represent MeshTV windows. The red "A" token represents the Powerwall active canvas. Mapping the "A" token sets the

Powerwall active canvas and causes all subsequent updates of MeshTV windows to be echoed to that canvas. Unmapping the "A" token causes updates of MeshTV windows to be echoed to the Powerwall canvases to which the MeshTV windows are mapped. All unmapped tokens numbered from 1 to 16 are implicitly mapped to Powerwall canvas one.

13.8 Token tray

The token tray is an area in the Powerwall window that contains unmapped tokens.

13.9 Mapping area

The mapping area is a picture of the Powerwall with the selected layout and token mapping. Depending on the layout, the mapping area is divided into one or more regions that correspond to Powerwall canvases. Use the middle mouse button to drag tokens from the token tray or other Powerwall canvases into the mapping area. Doing so maps a MeshTV window to a Powerwall canvas. Note that each Powerwall layout has its own mapping of MeshTV windows to Powerwall canvases.

13.10 Unmap all button

Press this button to unmap all MeshTV windows for the current Powerwall layout. After this button is pressed, any tokens in the mapping area are sent back to the token tray to indicate that they are unmapped. The tokens can then be remapped by dragging them onto the mapping area with the middle mouse button.

14.0 Species

14.1 Overview

Selecting this menu item pops up a window which allows you to manage which material species to use in the plots you create.

14.2 Species window

The **Species** window should look similar to Figure 6-17 when it pops up. The list of the **Active species** will contain entries if you have opened a file with materials containing material species.

The **Species** window allows you to manage which species in a material are plotted.

If you don't see a description for an item in this window, check in the Shared items section near the front of the chapter.

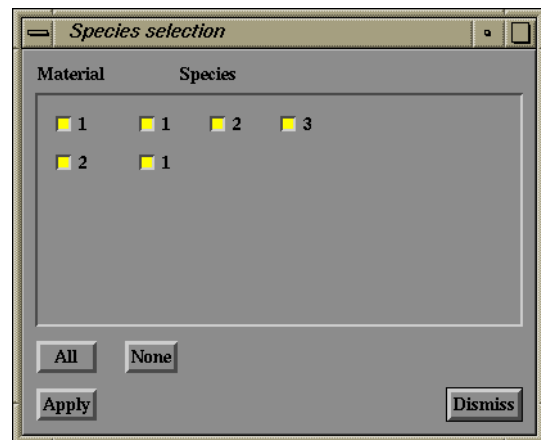


Figure 6-17: Species window

14.3 Active species list

This scrolled list contains all of the material species in the problem. A toggle button appears next to each material species number, and the state of the toggle indicates whether the material species is active or not. When the toggle is “pushed in”, the material species is active, and when it’s “popped out”, the material species is inactive. Only active species are plotted.

The first number in each row is the material itself, followed by the material’s species, so you can easily turn off entire materials, rather than just their species, via the first toggle in each row.

14.4 All button

Press this button to make all material species active. This is the default when MeshTV comes up. This is the quick way to make all your material species active, though you can also make species active by “pushing in” the toggle button next to the material species number.

14.5 None button

Press this button to make all material species inactive. You can use this when you want most species to be inactive. First make them all inactive and then select the ones you want by clicking on them.

Chapter 7

Extra Menu

1.0 Overview

This chapter covers the items found under the **Extra** menu at the top of the MeshTV **Main** window. The **Extra** menu, shown in Figure 7-1, contains the following items: **Command line**, **Command log**, and **Preferences**. Menu items with a ... in the name indicate that a window will open when the item is selected. The dotted line at the top of the menu denotes a "tear-off" menu, which means that when you click on the dotted line, the menu will "tear off" and stay posted until you close the window or press the ESC button while the window is selected.

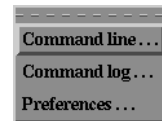


Figure 7-1: Extra

Extra menu

The **Extra** menu allows you to access the command line interface to MeshTV, view the log of commands already executed, or personalize certain MeshTV defaults.

Command line...

This button pulls up the command line interface window, which provides text-based interaction with MeshTV.

Command log...

This button brings up a log of commands that have already been executed.

Preferences...

This allows you to personalize MeshTV actions and protocols.

2.0 Command line

2.1 Overview

Selecting this menu item pops up the **Command line interface** window, which allows you to type commands to MeshTV.

2.2 Command line interface window

The **Command line interface** window is shown in Figure 7-2. The contents of the scrolled window might be different from the picture below when you bring up the window.

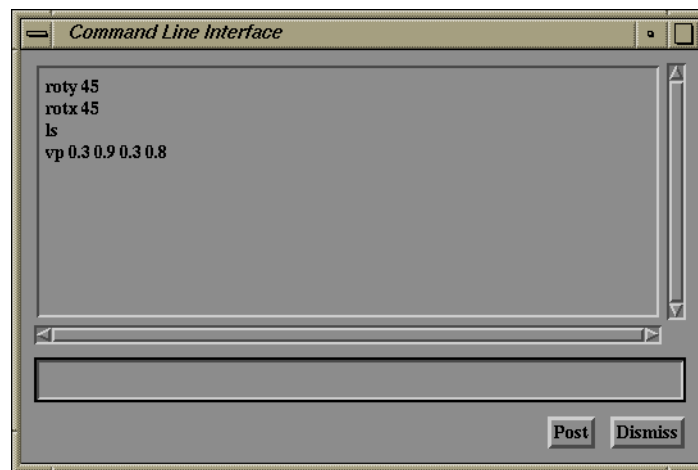


Figure 7-2: Command line interface window

Certain operations in MeshTV are either easier to execute with the command line interface, or are only accessible via typed commands. The **Command line interface** window allows the user to use typed commands. This window is usually posted to the MeshTV **Main** window in the **Notepad** area, but it can always be pulled up via the **Command line...** button in the **Extra** menu.

Note: Be careful when you type in commands via this text field, since you can use commands which might have undesired side-effects with the GUI. For a list of “safe” commands to use, see the *MeshTV Getting Started Manual*.

2.3 Command line scrolled window

This area contains the list of commands the user has previously typed. Users can “cut and paste” lines from this window to the **Command** text field to avoid retyping commands.

2.4 Command text field

Use this text field to type commands to MeshTV. Note that these commands will not be reflected by the windows in the GUI. For example, if you typed “printwin printer=myprinter”, the **Printer** text field in the **Set Print Options...** window would not be updated with “myprinter.”

Note: Be careful when you type in commands via this text field, since you can use commands which might have undesired side-effects with the GUI. For a list of “safe” commands to use, see the *MeshTV Getting Started Manual*.

2.5 Post/Unpost button

If you plan to use a window repeatedly, but you don’t want it cluttering up your screen, you might want to post it. When a window is posted, it appears in the **Notepad** area at the bottom of the MeshTV **Main** window, where it remains until it is unposted. Press this button to post the window. Once the window is posted, this button will change to **Unpost**.

2.6 Dismiss button

When you wish to close a window, press the **Dismiss** button.

3.0 Command log

3.1 Overview

Selecting this item pops up the **Command log** window, which allows you to see all the commands which have been executed during the current MeshTV session.

3.2 Command log window

The **Command log** window is shown in Figure 7-3. The contents of the scrolled window will probably differ from the contents in the picture below.

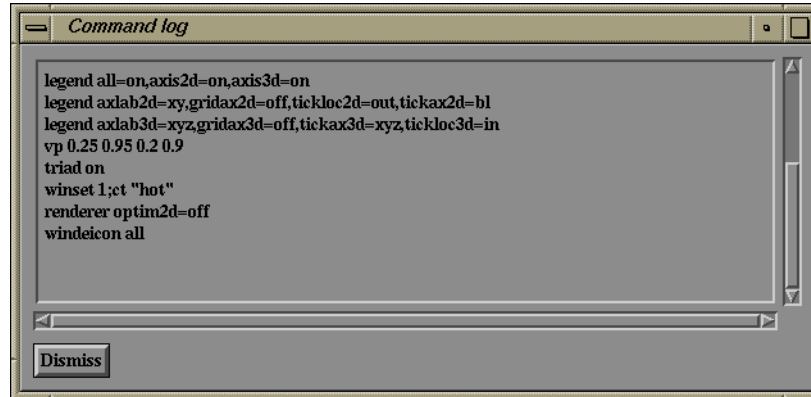


Figure 7-3: Command log window

This window lists all the commands issued either from the **Command line** window, or by the GUI itself. Users who want to construct their own log files to run MeshTV in batch mode might want to perform a given operation via the GUI and then use this window to see what the actual MeshTV commands are. Users can also look in the %meshtv.log file to see the commands. The %meshtv.log file is written to the directory from which MeshTV was run.

3.3 Dismiss

When you wish to close a window, press the **Dismiss** button.

4.0 Preferences

4.1 Overview

Selecting this item pops up a window which allows you to personalize your MeshTV session.

4.2 Preferences window

The **Preferences** window is shown in Figure 7-4. The options selected when you bring up the window might be different from the options selected in the picture below.

The **Preferences** window allows you to set your own options for various MeshTV functions. For example, if you prefer, you can decide to “turn off” informational messages so that you no longer see them. Any changes in this window remain in effect, till they are modified again. Modifications remain in effect for the rest of the current MeshTV session. To set permanent defaults for items in this window, you can make your changes to the window and then select the **Save settings** item in the **File** menu of the MeshTV **Main** window.

4.3 Information Display area

This area allows you to indicate which messages you would like to receive. For example, you can decide to see warnings but not messages to cut down on the number of informational windows you’ll see during your MeshTV session.

4.3.1 Display hints toggle button

You can turn this option on if you want to see hint messages. Hint messages contain helpful information on how to use MeshTV. When hints are enabled, moving the pointer over hint-enabled windows will display hints in a small raised window. Not all windows have hints.

4.3.2 Display messages toggle button

You can turn this option off if you don’t want to see informational messages. Messages contain helpful or confirmational information and as such require no action. For example, a message will pop up after you request **Getting Started...** from the **Help** pulldown menu, informing you that Netscape Navigator has been started.

4.3.3 Display warnings toggle button

You can turn this option off if you don’t want to see warnings. Warnings indicate that something is wrong, but not seriously wrong. For example, a warning pops up if you press

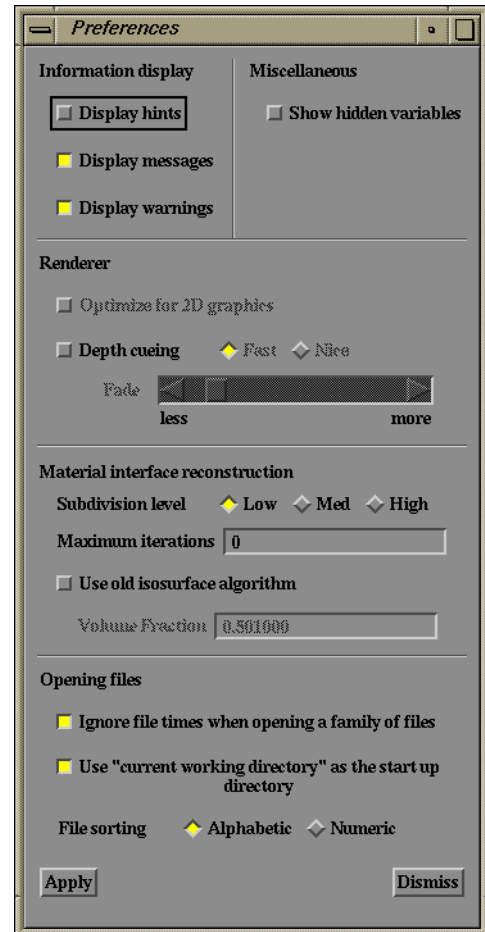


Figure 7-4: Preferences window

the **Apply** button in a plot attribute window, and there are no plots of that type (i.e., pressing the **Apply** button in the **Label plot attributes window**, and there are no label plots.)

4.4 Miscellaneous area

This area allows you to show hidden variables (variables which start with a dot.)

4.4.1 Show hidden variables toggle button

When displaying variables, MeshTV normally hides directories that start with a dot or are referenced by a multimesh object. When this option is set to on, all variables will be visible in the variable lists. Changing the setting affects only subsequently opened files, not currently open files. Most users never have occasion to turn this option on.

4.5 Renderer area

You can instruct MeshTV to customize the graphics renderer you're using. You can optimize graphics for two dimensional datasets by selecting the **Optimize for 2D graphics** toggle button, and you can fade details into the distance using the **Depth Cue** controls.

4.5.1 Optimize for 2D graphics toggle button

This option only applies if you are using the Software Renderer driver. Turning this on improves the speed and quality for all 2D plots except pseudocolor plots with nodal centering. This switches all 2D plot windows to the X11 driver from the Software Renderer, providing a significant speed increase.

4.5.2 Depth cueing controls

The **Depth cueing** toggle button option enables or disables depth cueing on three dimensional images. This blends objects into the background based on their distance from the viewer, and it can clarify details on complex objects.

When it is enabled, other controls are available. The quality selection radio buttons, **Fast** and **Nice**, hint to the renderer the method to use when applying depth cueing. Note, however, that when using the software renderer on very complex datasets, the **Nice** option may be faster. The **Fade** slider controls the amount of fading applied to the objects.

4.6 Material interface reconstruction controls

The options in this area deal with tuning the algorithm used for reconstructing material interfaces. This occurs in the material plots and when applying material selection to other

plots. When these options are changed, existing plots are not affected. Changes only affect future plots.

4.6.1 Subdivision level

This option selects the amount of subdivision to use. The lowest value implies no subdivision. This is the fastest option. The highest value can better handle zones with higher numbers of materials or complicated interfaces, but it takes longer and can introduce minor oscillations in the reconstructed surface.

4.6.2 Maximum iterations

This specifies the maximum number of iterations to use when attempting to make the volume fractions in the reconstruction represent the true volume fractions. It can be zero, implying no iteration, or a positive integer. Note that iteration can take a while in 3D, and that most of the accuracy is recovered after only a few iterations, so if you need more accuracy, start with an iteration count of 1 or 2, and work up from there.

4.6.3 Use old isosurface algorithm

When this option is on, any further material interface reconstruction will use the older algorithm. This algorithm is an isosurface-based approach which treats each material as the region where its volume fraction is greater than some value (typically 0.5). This algorithm is less accurate than the newer, probability-based algorithm, especially for clean zones and zones containing more than two materials. Note that the old algorithm is only available for 3D plots; this option does not affect 2D plots.

4.6.4 Volume fraction

The isosurface algorithm is not precise and can be tuned through its volume fraction parameter. When this value is reduced, material boundaries grow. When it is raised, materials encompass less.

4.7 Opening files area

The options in this area deal with what happens when files are opened. You can tell it to automatically reset attributes when you replace a file, ignore file times when reading in a family of files, and use the current working directory for your initial directory when selecting files.

4.7.1 Reset attributes on replace toggle button

When this option is on, plots are redrawn from data in the new file with most of the plot settings (e.g. contour levels, pseudocolor range, etc.) reset to values based on the new data

file. When this option is turned off, the attributes for the plots will stay exactly the same after the replace as before. When this option is on, pressing the **Draw** button on the MeshTV **Main** window without adding a plot causes the plots to be redrawn with the attributes reset.

4.7.2 Ignore file times toggle button

This option controls the reading of the times associated with each file in a family of files. If the option is off, the times inside each SILO file in the family are read when a member file is opened. If the option is set to on, the times inside each file will not be read when a family file member is opened. This greatly increases the speed of opening families of files. The drawback is that file times will no longer appear in the text field next to the animation slider on the main control panel. Also, the error checking insures that the times associated with each file in the family are monotonically increasing will be bypassed.

Unless you are running an animation which contains non-regular time intervals, you will probably want to set this option on, to avoid the slow down caused by reading file times. If you are running an animation which has constant time intervals, you don't need to read file times, because setting the frame to 3/4 through the animation will be both 3/4 in frames and 3/4 in time. However, if your files are irregularly spaced in time, setting the frame to 3/4 through the animation will be 3/4 in frames when you ignore files times, and 3/4 in time if you don't ignore file times while reading in files.

4.7.3 Use current working directory toggle button

When you select **Save settings** from the **File** menu in the MeshTV **Main** window, you can either save the directory path which is currently typed into the **Path** text field in the **Select files...** window, or you can instruct MeshTV to use whatever the current directory happens to be the next time MeshTV is run. MeshTV uses the current working directory when this option is on, else it uses the directory which was in the **Path** text field when **Save settings** was last selected.

4.8 Apply button

Pressing the **Apply** button applies changes you made in the window containing the button. If you don't want your changes applied, press the **Dismiss** button without pressing **Apply**.

4.9 Dismiss button

When you wish to close a window, press the **Dismiss** button. You should press the **Apply** button before you press the **Dismiss** button if you want your changes applied. If you don't want to apply your changes, pressing only the **Dismiss** button (without pressing **Apply**) will return you to previously applied values.

Chapter 8

Window Menu

1.0 Overview

This chapter covers items found under the **Window** menu at the top of the MeshTV **Main** window. These items are: **Add**, **Delete**, **Clear all**, **Flip Main window**, and **Layout**. The dotted line at the top of the menu denotes a "tear-off" menu, which means that when you click on the dotted line, the menu will "tear off" and stay posted until you close the window or press the ESC button while the window is selected. The **Window** menu window is shown in Figure 8-1.

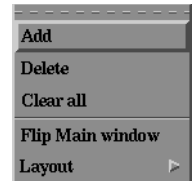


Figure 8-1: Window

Window menu

The **Window** menu allows you to add new visualization windows, delete existing windows, clear plots in all windows, flip the orientation of the MeshTV **Main** window, and organize your windows into specific layout patterns.

Add

This button adds a new visualization window.

Delete

This deletes the active visualization window.

Clear all

This removes the plots in all visualization windows.

Flip Main window

This entry allows you to flip the MeshTV **Main** window between a vertical and horizontal layout.

Layout

This item pops up a submenu that allows you to open different numbers of visualization windows. The size of the window depends on the layout chosen and the size of the computer screen.

2.0 Add

MeshTV starts with one window. To add other windows, select this menu item. Another window that is the same size as the last created MeshTV window will open. The number for this window is printed in the window titlebar. Once you have added more windows, either via this menu item, or via the **Layout** menu item, the **Active window** pulldown in the MeshTV **Main** window updates to include the new window, and the new window is made the active window.

3.0 Delete

Press this menu item to delete the active window. To determine the active window, look at the **Active window** pulldown in the MeshTV **Main** window. The number of the active window will be shown, and that is the window you will delete if you press this menu item. To delete a different window, use the **Active window** pulldown to change the active window and then hit the **Delete** menu item. Once the window has been deleted, window 1 becomes the active window, unless it was the window deleted, in which case the next lowest numbered window becomes the active window (i.e., window 2, if it exists, else window 3, etc.) Windows are not renumbered after a deletion.

4.0 Clear all

If you want to remove all the plots in all of MeshTV's output windows, select this menu item.

5.0 Flip Main window

The **Flip Main window** item allows you to flip between a vertical and horizontal layout for the MeshTV **Main** window. When the layout switches to vertical from horizontal, two windows are displayed, regardless of how many windows were shown when the **Main**

window was vertical. Likewise, when the layout switches from horizontal to vertical, one window is displayed, regardless of how many windows were shown when the **Main** window was horizontal. Note that no windows are deleted by this command, but a second window might be created.

6.0 Layout

6.1 Overview

This menu item offers six choices for the layout of MeshTV's windows: **1x1**, **2x1**, **2x2**, **2x4**, **3x3**, and **4x4**. These options will place the windows on an XY grid, so the window layout will be 1x1 (one window), 2x1 (two windows), 2x2 (four windows), 2x4 (eight windows), 3x3 (nine windows) or 4x4 (sixteen windows). The windows are sized to fit correctly onto the screen. If you have too few windows, extra windows are created to fill in. (For example, if you have three windows, and select **2x2**, another window is created.) If you have more windows than are needed for display, the extra windows are iconified. (For example, if you have sixteen windows, and then select **3x3**, the remaining seven are iconified.)

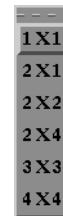


Figure 8-2: Layout

The dotted line at the top of the menu denotes a "tear-off" menu, which means that when you click on the dotted line, the menu will "tear off" and stay posted until you close the window or press the ESC button while the window is selected.

6.2 1x1

This layout choice places one visualization window on the screen. Other existing opened windows are iconified. This layout option produces the largest size windows. It is also the default when the **Main** window is vertically placed.

6.3 2x1

This layout choice places two visualization windows on the screen in a 2x1 grid. This layout works particularly well when the MeshTV **Main** window is in a horizontal layout, though it can also be used with a vertical layout. These windows are smaller than the 1x1 layout's windows and larger than the 3x3. If only one window is open when this option is chosen, another window is added. If there are more than two, the additional windows are iconified. It is also the default when the **Main** window is horizontally placed.

6.4 2x2

This layout choice places four visualization windows on the screen in a 2x2 grid. These windows are smaller than the 1x1 layout's windows and larger than the 3x3. If there are

fewer than four windows when this option is chosen, windows are added until four exist. If there are more than four, the additional windows are iconified.

6.5 2x4

This layout choice places eight visualization windows on the screen in a 2x4 grid. This layout works particularly well when the MeshTV **Main** window is in a horizontal layout, though it can also be used with a vertical layout. These windows are smaller than the 2x1 or 2x2 layout's windows and larger than the 3x3. If there are fewer than eight windows when this option is chosen, windows are added until eight exist. If there are more than eight, the additional windows are iconified.

6.6 3x3

This layout choice places nine visualization windows on the screen in a 3x3 grid. These windows are the third largest windows. If there are fewer than nine windows when this option is chosen, windows are added till nine exist. If there are more than nine, the additional windows are iconified.

6.7 4x4

This layout choice places sixteen visualization windows on the screen in a 4x4 grid. These windows are the smallest windows. If there are fewer than sixteen windows when this option is chosen, windows are added till sixteen exist. MeshTV supports a maximum of sixteen windows.

Chapter 9

Help Menu

1.0 Overview

This chapter covers items found under the **Help** menu at the top of the MeshTV **Main** window. The **Help** menu, shown in Figure 9-1, is used to find general information on MeshTV such as: help on how to use the GUI, copyright information, and updates on the latest versions of the program. This menu contains the following items: **On help**, **Getting Started**, **User's Manual**, **Command Line Interface**, **Release notes**, **Copyright**, and **Credits**. Menu items with a ... in the name indicate that a window will open when the item is selected. The dotted line at the top of the menu denotes a "tear-off" menu, which means that when you click on the dotted line, the menu will "tear off" and stay posted until you close the window or press the ESC button while the window is selected.

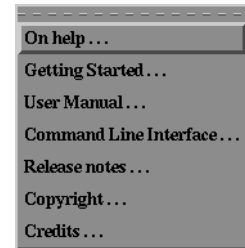


Figure 9-1: Help

Documentation on the **Error** window, which isn't accessible via the **Help** menu but does offer help, is located at the end of this chapter.

This section gives a quick description for each menu item. More detailed descriptions follow this section.

Help

The **Help** pulldown menu allows you to find out how to use the on-line help system, access MeshTV manuals, read about new bug fixes and functionality, read the copyright notice, and read our credits.

On help...

This pops up the **Help** window and details how to use the on-line help system.

Getting Started...

This brings up the *MeshTV Getting Started Manual* in Netscape Navigator™.

User's Manual...

This brings up the *MeshTV User's Manual* in Netscape Navigator™.

Command Line Interface...

This brings up the *MeshTV Command Line Interface Manual* in Netscape Navigator™.

Release notes...

This pops up a window which lists new functionality, bugs fixes, and workarounds for certain known bugs. These notes are organized by the MeshTV's release number. You can usually find the release number of the version you're running by looking at the shell window where you ran MeshTV. You should see something like "Running: meshtv3.0 -x meshtvx3.0", which indicates a release number of 3.0 in this example.

Copyright...

This pops up a window containing copyright information.

Credits...

This menu item pops up a window which gives credit where it's due.

2.0 On help

2.1 Overview

Selecting this item pops up the **Help** window. It provides information on how to use the MeshTV help system.

2.2 Help window

This window provides help for items (buttons, text fields, labels, radio buttons, etc.) in MeshTV. To access the window, simply position the cursor over an item and press the right mouse button. A relevant help message will display to the text area of this window.

Figure 9-2 shows an example of the **Help** window. This particular example would appear when you select **On help...** from the **Help** menu item in the MeshTV **Main** window. Note that the appearance of your window might differ slightly from the one shown here.

To get help on a particular item in any window in the MeshTV GUI, place the cursor over the item (button, text field, label, radio button, etc.) and press the right mouse button. The **Help** window will pop up, displaying any help available for that particular item.

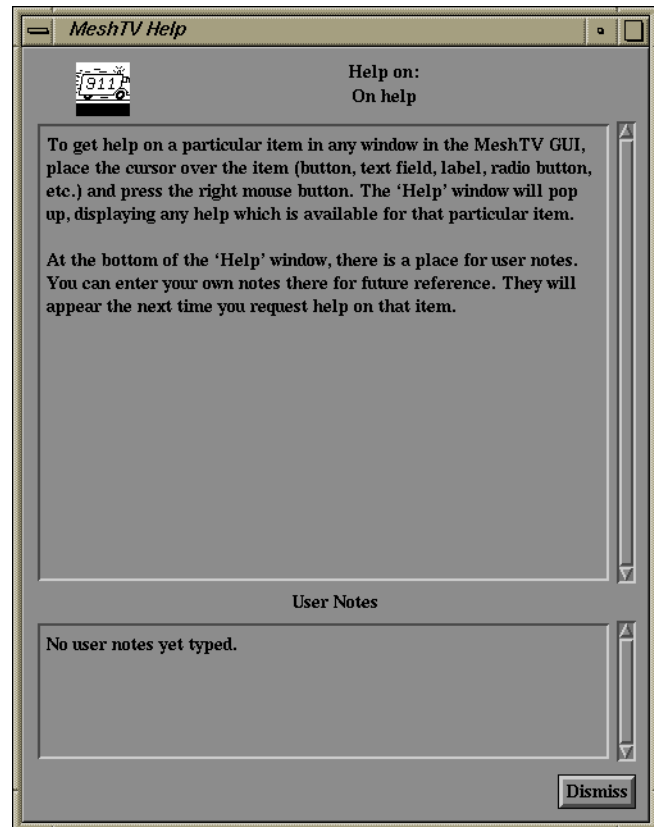


Figure 9-2: Help window

At the bottom of the **Help** window, there is a place for user notes. You can enter your own notes there for future reference. They will appear the next time you request help on that item. User help is saved automatically.

2.3 Help on

The text below **Help on:** shows the name of the item (button, text field, label, etc.) you selected for help. It helps you make sure you're getting information on the item you want and not for some other item.

2.4 Help text

Help information on the requested item (button, text field, label, etc.) displays here. To request information, place the cursor on the item for which you want help and right click.

2.5 User notes

This text area allows you to type your own help notes for an item. For example, if you request help on the **Raster postscript** printer type, you can include your own notes, which might detail the name of the printer you prefer. These notes are saved as you type them, and the next time you request help on the **Raster postscript** item, your notes will appear in this text area.

2.6 Dismiss button

When you wish to close a window, press the **Dismiss** button.

3.0 Getting Started

3.1 Overview

When you select this item, the *MeshTV Getting Started Manual* will display via Netscape NavigatorTM. If Navigator is already running, existing pages are replaced with the Getting Started pages. If Navigator is not running, the program is started.

4.0 User's Manual

4.1 Overview

When you select this item, the *MeshTV User's Manual* will display via Netscape NavigatorTM. If Navigator is already running, existing pages are replaced with the User's Manual pages. If Navigator is not running, the program is started.

5.0 Command Line Interface

5.1 Overview

When you select this item, the *MeshTV Command Line Interface Manual* will display via Netscape NavigatorTM. If Navigator is already running, existing pages are replaced with the Command Line Interface pages. If Navigator is not running, the program is started.

6.0 Release notes

6.1 Overview

This is the place to come to find the newest functionality, details on bug fixes, and the list of workarounds for known bugs. The appropriate areas are updated with each new release of MeshTV.

6.2 Release notes window

Figure 9-3 shows the window that pops up when you select the **Release notes...** item in the **Help** menu at the top of the MeshTV **Main** window. The actual contents of the window you see will probably differ from this picture.

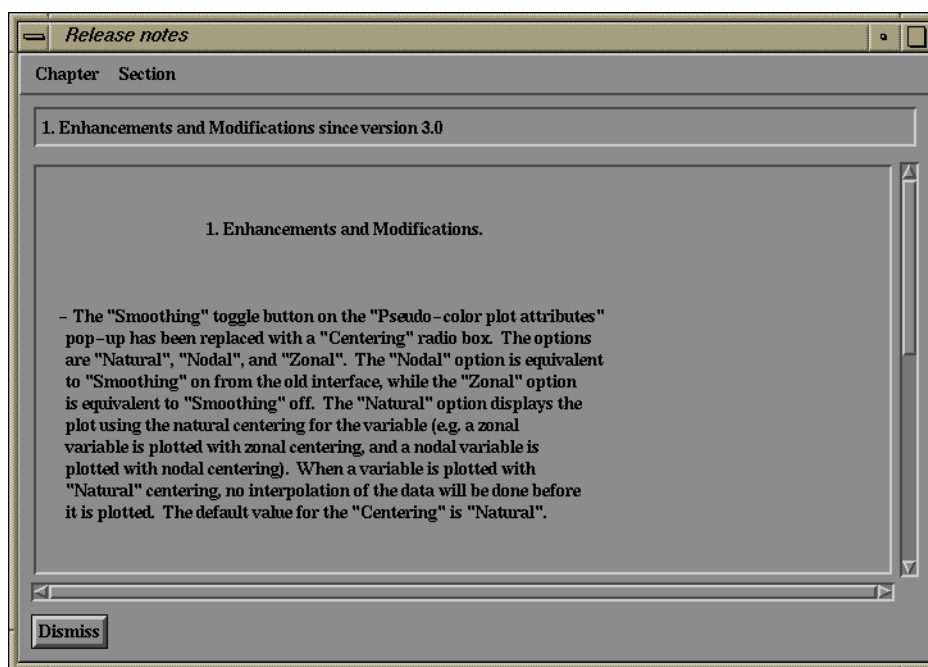


Figure 9-3: Release notes window

The **Release notes** window contains a scrolled window which displays information about MeshTV. The kind of information displayed is chosen via the **Chapter** and **Section** pulldown menus. The three kinds of information which can be displayed are **Enhancements and Modifications**, **Corrected Bugs**, and **Workarounds for Known Bugs**. This information can be displayed for the entire history of the code, or only for the history after a given version of the code. For example, you can choose to see corrected bugs for MeshTV after its version 3.0 release.

6.3 Display areas

The single line text area and the large scrolled window contain the output specified by the items in the **Chapter** and **Section** pulldown menus. The single-line text area indicates the kind of information displayed (**Enhancements and Modifications**, **Corrected Bugs**, or **Workarounds for Known Bugs**) and for which period of time the information is displayed (i.e., since version 3.0).

6.4 Chapter

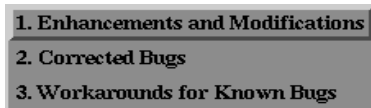


Figure 9-4: Chapter menu

The **Chapter** pulldown menu allows access to three kinds of information: **Enhancements and Modifications**, **Corrected Bugs**, and **Workarounds for Known Bugs**. These three elements work in concert with the **Section** pulldown menu by providing information only for specified versions of MeshTV.

6.4.1 Enhancements and Modifications

When this menu item is chosen, the display area in the **Release notes** window shows any enhancements or modifications to MeshTV since the release version specified by the **Section** pulldown. The chosen version is shown in the single line text area above the scrolled window display area. For example, it might read “Enhancements and Modifications since version 3.0.”

6.4.2 Corrected Bugs

When this menu item is chosen, the display area in the **Release notes** window shows any MeshTV bugs which have been corrected since the release version specified by the **Section** pulldown. The chosen version is shown in the single line text area above the scrolled window display area. For example, it might read “Corrected bugs since version 3.0.”

6.4.3 Workarounds for Known Bugs

Unfortunately we know of bugs in the code which have yet to be fixed. This section lists certain known bugs (and possible “workarounds” if they exist) in the display area in the **Release notes** window. Unlike the **Enhancement and Modifications** and **Corrected Bugs** items, **Workarounds for Known Bugs** does not rely on the **Section** pulldown since it lists currently known bugs rather than bugs associated with a given version of the code.

6.5 Section

The **Section** menu allows you to select how much information you wish to receive on **Enhancements and Modifications** or **Corrected Bugs**. The items in this menu indicate different releases of MeshTV. If you want to see all changes after a given release, select the item which indicates that release. For example, if you want all corrected bugs since version 3.0, you select **Corrected Bugs** in the **Chapter** menu and **since version 3.0** in the **Section** menu.

6.6 Dismiss button

When you wish to close a window, press the **Dismiss** button.

7.0 Copyright

7.1 Overview

MeshTV is copyrighted by the Department of Energy. The copyright notice details the legal information and disclaimers. Read this if you're experiencing insomnia, or if you need to find out legal information about MeshTV.

7.2 Copyright window

Figure 9-5 shows an example of the copyright window.

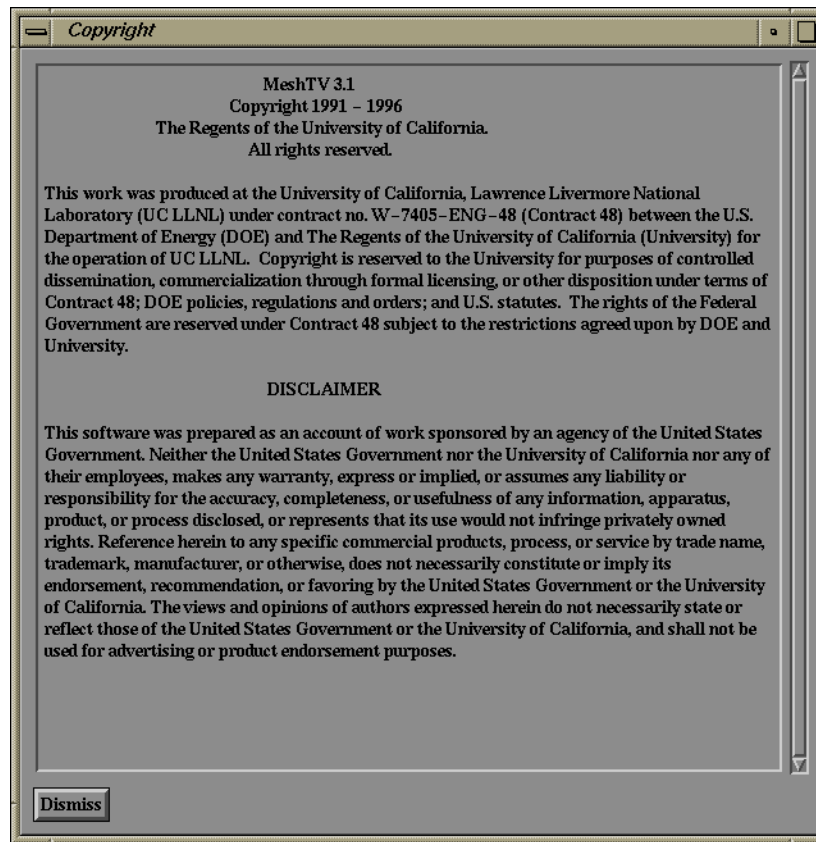


Figure 9-5: Copyright window

This window displays the copyright information for MeshTV. It details the auspices under which the work was performed and reserves the rights for the program to the University of California and the Federal Government.

7.3 Dismiss button

When you wish to close a window, press the **Dismiss** button.

8.0 Credits

8.1 Overview

This window gives credit to Lawrence Livermore National Laboratory, who sponsored the MeshTV project.

8.2 Credits window

Figure 9-6 shows the Credits window.

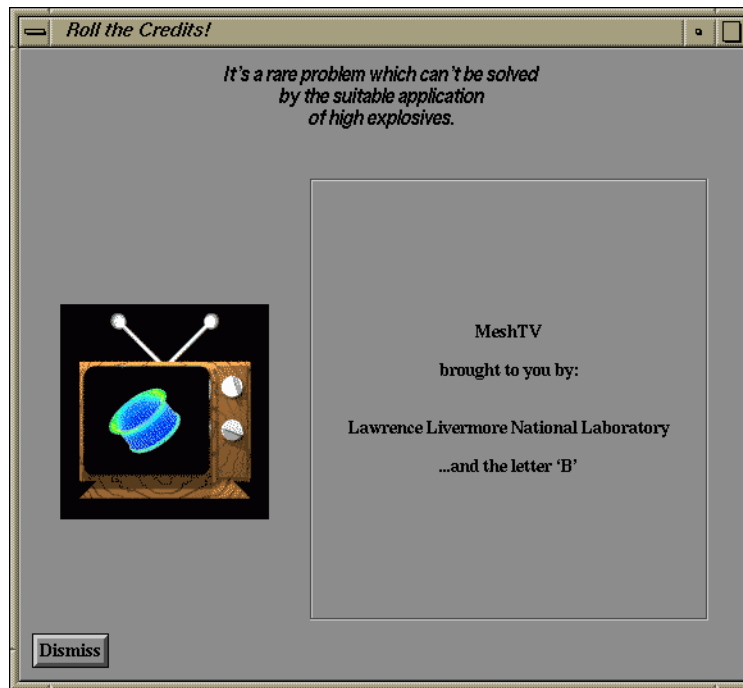


Figure 9-6: Credits window

This window shows a nifty picture, credits Lawrence Livermore National Laboratory, and displays the motto of the MeshTV development team. It is brought to you by LLNL and the letter 'B.'

8.3 Dismiss button

When you wish to close a window, press the **Dismiss** button.

9.0 Information window

9.1 Information window



Figure 9-7: Error window

Read release notes button will appear in the information window. If you press this button, the release notes window will open, allowing you to read the release notes for the new version of the code.

This window pops up to report error messages, warnings, and notifications.

Figure 9-7 shows the **Information** window

9.2 Dismiss button

When you wish to close a window, press the **Dismiss** button.

9.3 Read release notes button

When MeshTV has been updated, the **Read release notes** button will appear in the information window. If you press this button, the release notes window will open, allowing you to read the release notes for the new version of the code.

10.0 Query window

10.1 Information window



Figure 9-8: Query window

This window pops up to report a problem and as a yes/no question to handle the problem. When this window is up, you can't do anything else in the Graphical User Interface. You must answer the question before continuing.

Figure 9-8 shows the **Query** window asking the user whether to apply an orthogonal slice operator.

Chapter 10

Visualization Windows

1.0 Overview

This chapter covers the graphical controls in the visualization window. These are the Popup menu and the Toolbar. A visualization window is shown in Figure 10-1.

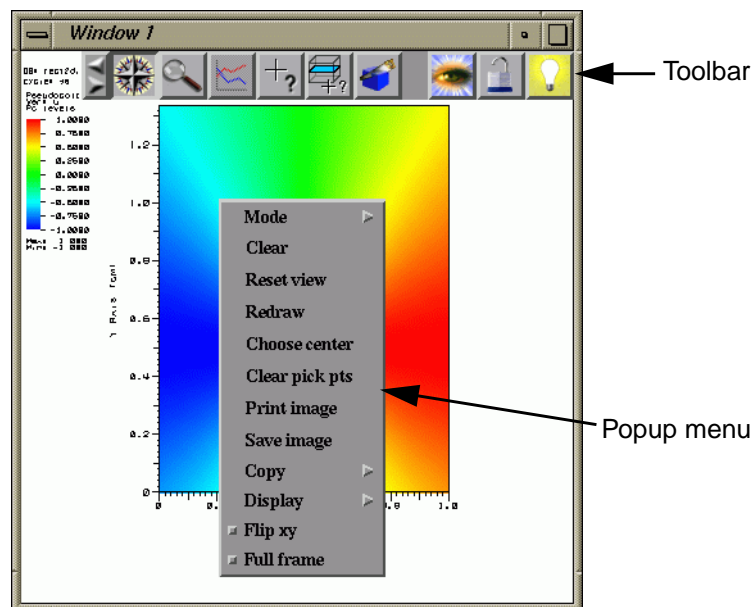


Figure 10-1: A Visualization window

2.0 Window modes

To understand the controls for a visualization window, you must first understand the use of window modes. A visualization window has six modes: Navigate, Zoom, Lineout, Pick, Slice-pick, and Slice.

2.1 Navigate mode

When the window is in Navigate mode, you can move the dataset around, rotating (3D datasets only), panning, and zooming in and out.

- **Rotation**

This only applies to a 3D dataset. Press and hold the left mouse button in the visualization window. Move the mouse, and the dataset rotates. Release the mouse button when you have the view you want.

- **Panning**

For a 2D dataset, press and hold the left mouse button. Move the mouse, and the dataset will shift, following the mouse. Release the mouse button when you reach the desired view.

Panning a 3D dataset is identical to panning a 2D dataset, except that you must hold down the Alt key before pressing the left mouse button.

- **Zooming**

For both kinds of datasets, press and hold the middle mouse button to incrementally zoom into a dataset. Holding the Alt key while pressing the middle mouse button incrementally zooms out of a dataset.

2.2 Zoom mode

When the window is in Zoom mode, you can draw a box around the area of the dataset you wish to zoom into. Press and hold the left mouse button at one corner of the area you wish to zoom into. As you move the mouse, the zoom box sweeps out from the first corner. Release the mouse button when the zoom box covers the desired area.

2.3 Lineout mode

This mode is only available for 2D datasets. When the window is in Lineout mode, you can draw a line across a dataset and have the scalar variable data along that line display as a curve to another window. To draw a line from which to extract data, press and hold the left mouse button at one of the endpoints of the line. Move the mouse to extend the line from the first point. Release the mouse button at the second endpoint. A line is drawn between the two points.

Depending on the type of lineout you have chosen, using the GUI (Graphical User Interface) or the “`refl`” CLI (Command Line Interface) command, the line will either be straight or will follow the contours of the mesh.

Once the line is drawn, the scalar variable data along that line drawn as a curve, or “1D plot” to another window.

2.4 Pick mode

When the window is in Pick mode, you can extract information about data at a particular point in the window. Click with the left mouse button on the point you wish to extract information about.

2.5 Slice pick mode

This mode only is available when interacting with 3D datasets. When you enter this mode, the view of the dataset changes. Rather than seeing the full 3D dataset, you will instead see a slice of the dataset. You can slide this slice backward and forward using the left mouse button. Press and hold the left mouse button to start sliding the slice. Move the mouse up or down until the slice is in the desired location. Release the mouse button. You can repeat these actions to move the slice until you have it where you’d like it. Once the slice is in the desired location, click with the left mouse button on the point on the slice where you wish to extract information.

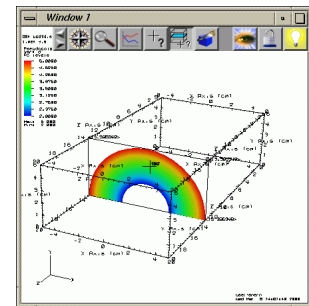


Figure 10-2: 3D pick

The slice is always aligned with the coordinate axes. The axis that points closest to “out of the screen” is the one chosen to slice along. Thus, if you wish to slice along a different axis when doing 3D pick, first rotate the plot using Navigate mode. Once the dataset is facing the right direction, change to 3D pick mode.

2.6 Slice mode

This mode is available when interacting with 3D datasets. Like slice pick mode, when you enter slice mode, the view will change to a slice of the dataset. This slice plane can be rotated and translated to any position in the dataset. Press and hold the left mouse button while moving the mouse to rotate the slice plane about its origin. This will display a cyan slice plane with a large black normal arrow. The plane and arrow are visible for the duration of the rotation operation and are displayed so you can see the orientation of the slice plane as it changes. Once the left mouse button is released, the dataset is re-sliced and

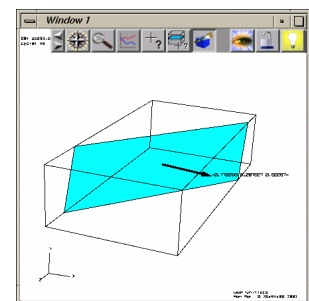


Figure 10-3: Slice mode

the new slice is drawn in the visualization window. Press and hold the middle mouse button while moving the mouse to move the slice plane along its normal vector. This will move the slice plane along an axis that is perpendicular to the slice plane. The slice plane's origin can be changed by holding down the Shift key while holding down the left mouse button. With this combination of keyboard keys and mouse buttons held down, the origin of the slice plane can be moved side to side or up and down when the mouse is moved. The slice plane origin can be moved to and fro by holding down the Shift key and the middle mouse button while moving the mouse.

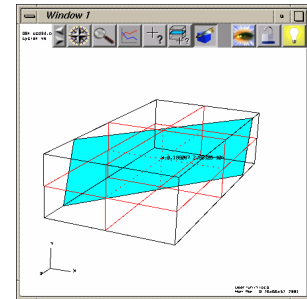


Figure 10-4: Slice mode

3.0 The Popup menu

3.1 Overview

The Popup menu contains menu items for various operations that apply to the window. There are also toggle buttons to set state information. Access the menu by pressing and holding down the right mouse button. Select the desired item, then release the mouse button.

The items in the Popup menu are: the **Mode** menu, **Clear**, **Reset view**, **Redraw**, **Choose center**, **Print image**, **Save image**, the **Copy** menu, the **Display** menu, the **Flip xy** toggle, and the **Full frame** toggle.

This section gives a quick description of each item. More detailed descriptions follow this section.

Mode menu

The **Mode** menu contains radio buttons for each of the six menu modes. Selecting a button switches the window to that mode.

Clear

Selecting this menu item clears all plots from the window. They can be drawn again by pressing the **Draw** button in the GUI or reissuing the CLI “plot” commands.

Reset view

Selecting this menu item resets the viewpoint and zoom level back to the default view. The default view is the one that is used when a dataset is first displayed in a window.

Redraw

Selecting this menu item redraws all of the graphics in the window.

Choose center

Selecting this menu item allows you to choose the center of rotation for 3D images.

Clear pick pts

Selecting this menu item clears any pick labels that have been attached to the visualization window's plots.

Print image

Selecting this menu item sends the image in the window to a printer using the current printer settings.

Save image

Selecting this menu item saves the image in the window to a file using the current save window settings.

Copy menu

Use the items in this menu to copy window attributes from one window to another.

Display menu

Use the items in this menu to change the look and feel of the graphics in the window.

Flip xy toggle

Turning this toggle on transposes the X and Y axes of 2D plots. Turning the toggle off reverses the action. The toggle only applies to 2D datasets.

Full frame toggle

Turning this toggle on causes 2D plots to fill up the visualization window, stretching the axes to do so. This can be useful when a dataset is long and thin and difficult to see with

the normal view. Turning the toggle off restores the plot to the normal view. The toggle only affects 2D datasets.

3.2 Mode menu

This menu is shown in Figure 10-5. It contains four items: **Zoom**, **Navigate**, **Line-out**, **Pick**, and **Slice pick**.

These menu items correspond to window modes, as described in Section 2.0. A yellow “pushed-in” diamond highlights the mode that the window is in, as illustrated in the figure by the “pushed-in” yellow diamond next to **Navigate**.

Selecting one of these menu items switches the window to that mode. If a particular mode is not applicable (e.g. Line-out for a 3D window), it will be grayed out and you will not be able to select it.

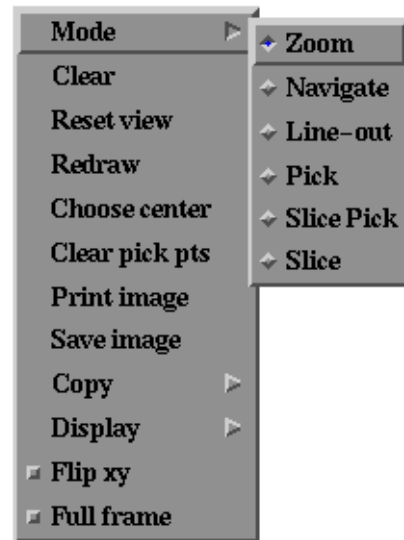


Figure 10-5: The mode menu

These menu items are equivalent to entering the CLI commands: “winmode zoom”, “winmode navigate”, “winmode lineout”, “winmode pick”, “winmode slicepick”, and “winmode slice” respectively.

3.3 Clear

This menu item removes all plots displayed in the window. Annotations and legends are also removed, leaving the window blank. To restore the plots, either press the **Draw** button in the GUI or enter “plot” commands in the CLI.

This menu item is equivalent to entering the CLI command “clear”.

3.4 Reset view

This menu item resets the viewpoint and zoom level back to that seen when the dataset was first displayed in a window. You might use this when the view is confusing, and you wish to go back to where you started.

This menu item is equivalent to entering the CLI command “reset win”.

3.5 Redraw

This menu item redraws all of the graphics in the window.

This can be especially useful when you are displaying windows to a tiled display, or powerwall. Windows may be echoed on a tiled display using the **Powerwall** controls in the GUI or the “powerwall” command in the CLI. When you want to redraw a window on the tiled display, use this command.

This menu item is equivalent to entering the CLI command “redraw”.

3.6 Choose center

This menu item allows you to select the center of rotation for 3D images.

Selecting this menu item causes the cursor to change shape to a plus sign. You may then press and release any mouse button over the point you want for the new center of rotation. The new center of rotation will correspond to the x, y, and z position of the point on the front surface under the mouse. The new center of rotation will be displayed in the *Output* window. The visualization window will go into navigate mode.

This menu item is equivalent to entering the CLI command “center3 pick”.

3.7 Clear pick pts

This menu item allows you to clear pick labels that have been added to your plots as a result of having been in pick mode.

This menu item is equivalent to entering the CLI command “delete pick”.

3.8 Print image

This menu item sends the image displayed in the window to a printer. The printer must already be set up using the **Set print options...** menu item in the GUI or the options of the “printwin” command in the CLI.

This menu item is equivalent to selecting **Print window** in the GUI’s **File** menu or entering “printwin” in the CLI.

3.9 Save image

This menu item saves the image displayed in the window to a file. The save options must already be set up using the **Set save options...** menu item in the GUI or the options of the “savewin” command in the CLI.

This menu item is equivalent to selecting **Save window** in the GUI’s **File** menu or entering “savewin” in the CLI.

3.10 Copy menu

This menu is shown in Figure 10-6. It contains five items: **View from**, **Lighting from**, **Annotations from**, **Palette from**, and **Everything from**. Each of the items is a menu with a submenu containing a list of window numbers.

Each menu item allows you to copy attributes to the current window from another window. Each submenu provides a list of appropriate source windows to copy from.

3.10.1 View from

This menu lets you copy the view attributes from another window to the current window. The view attributes consist of the direction of the viewpoint and the zoom level.

Copy the view attributes when you would like one window's view of a dataset to be identical to that of another window.

This operation is equivalent to entering the CLI command “`copyatt source_win dest_win view`”, where *source_win* and *dest_win* are the window numbers of the source window and destination, respectively.

3.10.2 Lighting from

This menu lets you copy the lighting from another window to the current window. The lighting attributes consist of the definition of all eight lights, whether they are on, where they point, their colors, their types, and their intensities.

This operation is equivalent to entering the CLI command “`copyatt source_win dest_win lighting`”, where *source_win* and *dest_win* are the window numbers of the source window and destination, respectively.

3.10.3 Annotations from

This menu lets you copy the annotation attributes from another window to the current window. The annotation attributes consist of all settings in the GUI's **Annotation** window. It encompasses the definition of legends, database and user information, 2D and 3D axes, and banners.

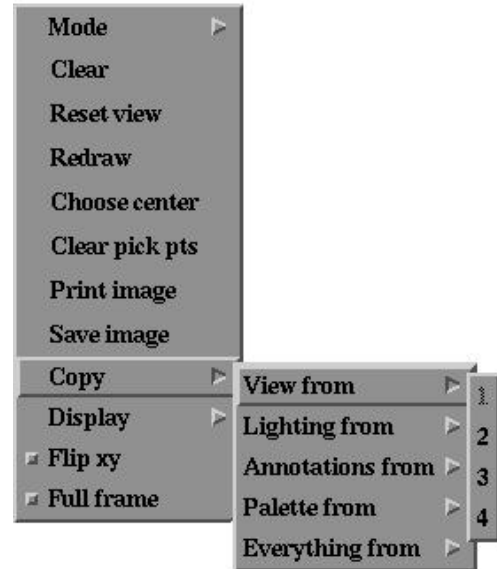


Figure 10-6: The Copy menu

This operation is equivalent to entering the CLI command “`copyatt source_win dest_win annotation`”, where *source_win* and *dest_win* are the window numbers of the source window and destination, respectively.

3.10.4 Palette from

This menu lets you copy the palette from another window to the current window. The palette is the definition of all of the discrete colors used in plots.

Copying the palette is useful when you wish the material colors in one window to be identical to those in another window.

This operation is equivalent to entering the CLI command “`copyatt source_win dest_win palette`”, where *source_win* and *dest_win* are the window numbers of the source window and destination, respectively.

3.10.5 Everything from

This menu performs a combination of the other copy menus. Copying “everything” from one window to another is equivalent to copying the view, lighting, annotation, and palette attributes.

This operation is equivalent to entering the CLI command “`copyatt source_win dest_win all`” or “`copyatt source_win dest_win view annotation lighting palette`”, where *source_win* and *dest_win* are the window numbers of the source window and destination, respectively.

3.11 Display menu

This menu is shown in Figure 10-7. It contains six items: **Invert background**, **Navigate bbox**, **Display toolbar**, **Perspective**, **Spin**, and **Stereo**.

3.11.1 Invert background

Selecting this menu item inverts the background and foreground colors of the window.

This operation is equivalent to entering the CLI command “invert”.

3.11.2 Navigate bbox

This toggle changes how 3D datasets appear when rotated. Normally, datasets redraw themselves as you rotate them. If the **Navigate bbox** toggle is turned on, however, the dataset is replaced by its bounding box while being rotated. Showing only the bounding box when rotating greatly improves interaction when you’re working with a large dataset or over a slow graphics connection. Once you stop rotating the dataset, it is redrawn normally in the new position.

This operation is equivalent to entering the CLI command “navigate bbox” and “navigate normal”.

3.11.3 Display toolbar

This toggle lets you remove the toolbar from the top of the visualization window. Turning the toggle back on displays the toolbar again.

There is no CLI equivalent for this operation.

3.11.4 Perspective

This toggle lets you change whether perspective is on. If perspective is on, the 3D graphics in the window are displayed in a “realistic” way, with graphics growing smaller with increasing distance. If perspective is off, the 3D graphics in the window are displayed using an “orthographic” projection, where distance has no effect on the size of the graphics. For an example, see Figure 10-8.

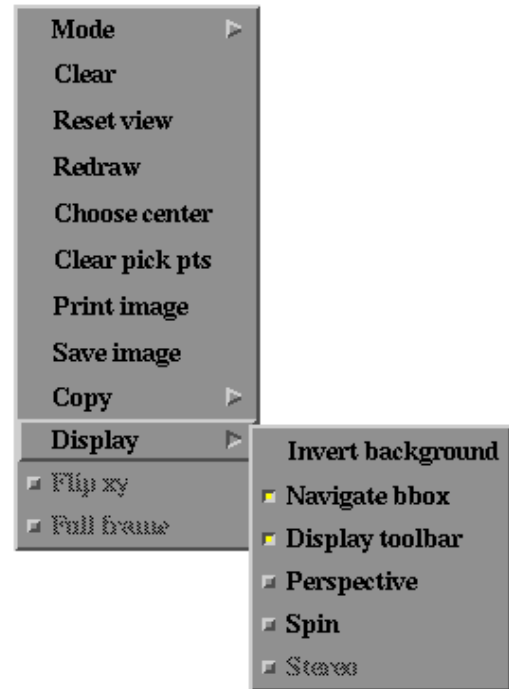


Figure 10-7: The Display menu

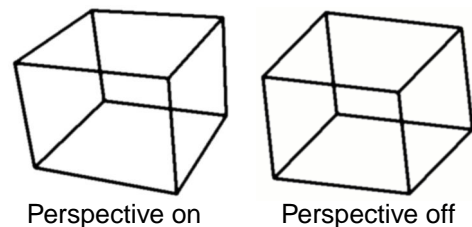


Figure 10-8: Perspective examples

This operation is equivalent to entering the CLI command “perspective on” and “perspective off”.

3.11.5 Spin

This toggle changes the way objects rotate in the window. Normally, rotation stops if you release the mouse button while a dataset is rotating. If this toggle is turned on, however, the dataset will continue rotating after the mouse button is released.

This operation is equivalent to entering the CLI command “spinmode on” and “spinmode off”.

3.11.6 Stereo

This toggle lets you create a stereo view of your dataset. It is supported only on certain platforms, and only if you also have special “3D glasses” (available from your computer vendor). Turning this toggle on sends slightly different views of the window to each eye. This yields a “true 3D” view of the dataset.

This operation is equivalent to entering the CLI command “stereo on” and “stereo off”.

3.12 Flip xy toggle

Selecting this toggle changes the orientation of 2D plots. Normally, the X axis is horizontal, and the Y axis is vertical. If this toggle is turned on, the axes are flipped so the X axis is vertical and the Y axis is horizontal. See Figure 10-9 for an example.

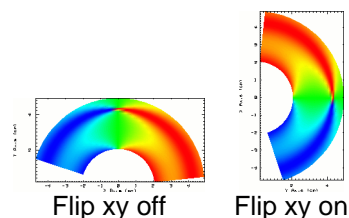


Figure 10-9: Flip xy example

This operation is equivalent to entering the CLI command “flipxy on” and “flipxy off”.

3.13 Full frame toggle

This toggle changes the scale of 2D plots. Normally, the same scale is used for both the X and Y axes. If this toggle is turned on, the scales for the two axes are calculated independently to make the plot square. If your dataset is long and thin and difficult to see, using fullframe can help by expanding the dataset along its shorter axis. See Figure 10-10 for an example.

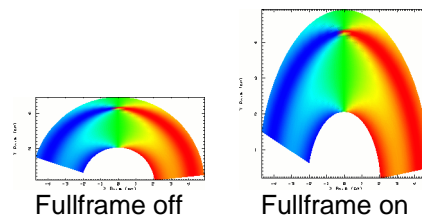


Figure 10-10: Fullframe example

This operation is equivalent to entering the CLI command “fullframe on” and “fullframe off”.

4.0 The toolbar

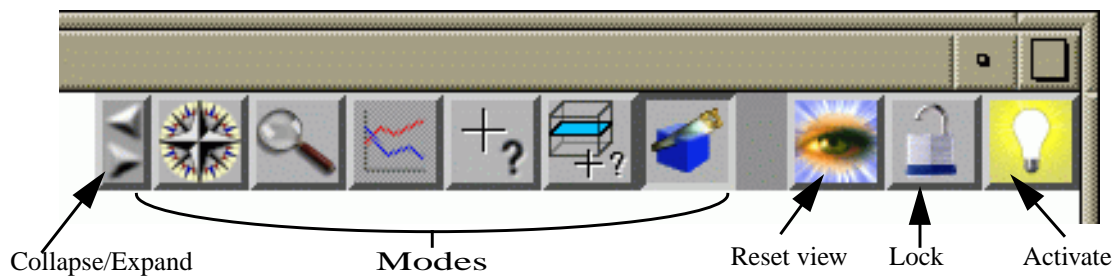


Figure 10-11: The toolbar

4.1 Overview

The toolbar contains buttons that perform various operations in the visualization window. Some of these buttons also show window state. There are three sections of the toolbar, as seen above in Figure 10-11: The Collapse/Expand button, the Mode buttons, the Reset view button, the Lock button, and the Activate button. If you put your mouse over a button and leave it there for a moment, a small window, or “tooltip”, appears, describing the function of that button.

This section gives a quick description of each toolbar item. More detailed descriptions follow this section.

Collapse/Expand button

This button makes the toolbar “slide” into the window, hiding the other buttons. Pressing it again expands the toolbar.

Mode buttons

These buttons show the window’s current mode, and switch the window to the given mode when you click on them.

Reset view button

Pressing this button resets the viewpoint and zoom level back to the default view. The default view is the one that is used when a dataset is first displayed in a window.

Lock button

Pressing this button toggles a window’s “lock view” state.

Activate button

Pressing this button causes an inactive window to become active.

4.2 Collapse/Expand button

This button allows you to “collapse” the toolbar into a tiny space as seen in Figure 10-12. This allows easy access to the toolbar without it taking up much screen space. Pressing the button again makes the toolbar expand.

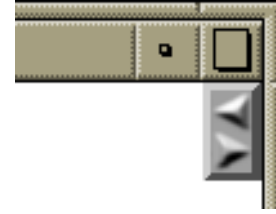


Figure 10-12: Collapsed toolbar

4.3 Mode buttons

These buttons correspond to window modes, as described in Section 2.0. A “pushed-in” button indicates the mode that the window is currently in. For example, as illustrated in Figure 10-11, the “pushed-in” compass rose indicates that the window is in Navigate mode.

The buttons, from left to right, are: Navigate mode, Zoom mode, Lineout mode, Pick Mode, Slice pick, and Slice mode.

Selecting one of these buttons switches the window to that mode. If a particular mode is not applicable (e.g. Lineout for a 3D window), the button will be grayed out and you will not be able to push it.

These buttons are equivalent to entering the CLI commands: “winmode zoom”, “winmode navigate”, “winmode lineout”, “winmode pick”, “winmode slicepick”, and “winmode slice” respectively.

4.4 Reset view button

This button resets the viewpoint and zoom level back to that seen when the dataset was first displayed in a window. You might use this when the view is confusing, and you wish to go back to where you started.

Pushing this button is equivalent to entering the CLI command “reset win”.

4.5 Lock button

This button toggles the window’s “lock view” state. If the window view is unlocked (the default), the button appears as an unlocked padlock. If the window’s view is locked, the button appears as a locked padlock with a small eye to the upper left. All, some, or no windows may have their view locked at any time. See Figure 10-13 to see how this button can appear.

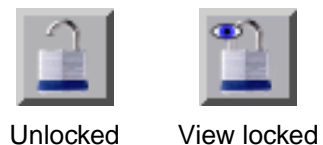


Figure 10-13: The lock button

When a window has its view locked, its view attributes (See Section 3.10.1) are “tied” to other windows that have their view locked. Thus, if you change the view (through a rotation or pan, for example) of a window whose view is locked, all other windows that are locked by view will reflect the effects of your manipulation. This keeps multiple windows in sync, allowing you to focus on the same part of a dataset.

To lock a window’s view, push the lock button. To unlock a window’s view, push the button again. The lock button thus acts as a toggle.

When you lock a window’s view, the window replaces its view with that from the other windows that are locked by view. For example, when windows 1 and 2 are already locked, locking window 3’s view copies the view from windows 1 and 2 to window 3.

Pressing this button is equivalent to entering the CLI command “lock view *num*” and “unlock view *num*”, where *num* is the window’s number.

4.6 Activate button

This button displays a window’s “active” status. Only one window may be active at any time. It is called the “active window”. The active window’s Activate button appears as a lit light bulb. All other windows have unlit light bulbs for their Activate button. See Figure 10-14 to see how this button can appear.



Figure 10-14: The Activate button

Clicking on an unlit light bulb causes that window to become the active window. The previously active window becomes inactive. Clicking a lit light bulb has no effect.

When running MeshTV from the CLI, these buttons provide a quick visual indication of which window is currently active. The button is even more useful when MeshTV is run with the GUI. While the GUI has a menu for changing the active window, this button provides a faster way of doing so. For example, if you want to quickly see all of the plots in the four windows you currently have open, you can click on each window’s Activate button. As soon as each window becomes active, the GUI updates with that window’s information.

Pressing this button is equivalent to entering the CLI command “winset *num*”, where *num* is the window’s number.

Chapter 11

Glossary

1.0 Table of definitions

Active window	The visualization window to which new plots or operations will display or occur.
Animation	MeshTV can animate files which are members of a file family. An animation involves showing pictures one after another to make a “movie.” One picture comes from each file.
Background color	The color of the background in a visualization window.
Block	The fundamental building block of a mesh. It defines the nodal coordinates of one contiguous section of a mesh, and is also known as a mesh block.
Clean zone	A zone which contains only one material.
CLI	Command Line Interface. Pronounced one letter at a time. See Command Line Interface.
Command Line Interface	A method of interacting with MeshTV by issuing commands. You can find a list of commands which MeshTV understands in the MeshTV Command Line Interface Manual. Using a command line interface requires more typing, but is often quicker than a GUI for an experienced user.
Computational mesh	See Mesh.

Data extents	Generally, the absolute minimum and maximum of a variable. However, the user can set the data extents that a particular plot will view, in which case, the data extents are the minimum and maximum user-specified values of a variable that are considered for visualization. Different plots may do different things with values outside of these data limits. For example, one plot may decide not to draw values outside of the range. Others may treat them as values at the ends of the range. Unless the user explicitly sets the minimum and maximum, however, the data extents are the absolute minimum and maximum of the variable.
Data limits	See Data extents.
Depth Cueing	The coloring of an graphical item based on its distance from the viewer, giving an impression of depth. In MeshTV, when depth cueing is enabled, items farther away are blended with the background color.
Family	See File families.
File families	A grouping of files which adhere to the following naming convention: All files share the same root name and differ in appended numbers. For example, the files “rect0001.silo” and “rect0002.silo” comprise a family of files. MeshTV can animate families of files.
Foreground color	The color of the foreground in a visualization window.
Graphical User Interface	A method of interacting with MeshTV by using buttons, menus and lists. This minimizes the amount of typing users must perform to accomplish goals, but tends to be slower than a CLI would be for more experienced users.
Grayed out	In MeshTV’s GUI, when an item (called a widget) like a button or label is unavailable for manipulation, its text will appear slightly fuzzy or “grayed out.”
GUI	Graphical User Interface. Pronounced “Gooley.” See Graphical User Interface.
Isosurface	A surface connecting points of constant value for a variable.
Line-out	A curve, or x/y plot, of the values of a variable along a line or other curve.
Material	A physical material, like hydrogen or glass, being modeled in a computer simulation. A material is comprised of one or more material species.
Material species	A single component of a material. A material can contain one or more material species. For example, the material Air contains the species Oxygen and the species Nitrogen. The portion of the material in a zone for a particular material species is measured by mass fraction, since the species are assumed to be distributed evenly throughout the material in the zone.
Mesh	Composed of one or more blocks.
Mixed material	A zone which contains multiple materials.
Mixed zone	A zone which contains multiple materials. Also called a mixed material zone.
Multi-species material	A material with more than one material species.
Node	A mathematical point. The fundamental building block of mesh or zone.
Notepad	An area at the bottom of MeshTV’s Main window in the GUI. This area allows you to post windows for future access.

Operator	A “filter” that can be applied to the data. Operators manipulate the data so that various features can be seen. For example, one such operator might slice a 3D dataset so that 2D information can be seen. Another operator might reduce the dataset to allow for easier manipulation.
Plot	A graphical representation of data. Examples include graphs of variable values, materials, or the computational mesh.
Pointmesh	A mesh consisting of set of locations, or points, in space. These nodes are not connected.
Post	To place the items in a window into the Notepad in MeshTV’s Main window.
PostScript	A format for saving or printing image data.
Pseudocolor	A plot which maps a variable’s values to a corresponding color from a color table. This allows users quickly grasp the range of numerical values of a given variable over a section of the data.
Raster PostScript	A raster format for saving or printing image data.
RGB	SGI’s Red Green Blue format. A format for saving image data.
Single-species material	A material with one material species.
Quadmesh	Quadrilateral mesh. A mesh which contains four nodes per zone in 2D and eight nodes per zone (four nodes per zone face) in 3D. These meshes are logically rectangularly connected.
SILO	A library developed at Lawrence Livermore National Laboratory to handle scientific database issues, such as cross-platform compatibility, mesh, material, and variable data structures, and multidimensional arrays. MeshTV can read and display information from SILO files.
TIFF	Tagged Image File Format. A format for saving image data.
Ucdmesh	Unstructured mesh. A general mesh representation composed of an arbitrary list of zones of arbitrary sizes and shapes. A quadmesh can be represented as a ucdmesh, but not the other way around, since quadmeshes have an implied rectangular connectivity, and ucdmeshes have no implied connectivity.
Unpost	To move the items in a window in the Notepad in MeshTV’s Main window back into their own window.
Variable	Data which are associated with a computational mesh. Variables usually represent values of some physics quantity, like pressure or density. Values are located either at the mesh nodes or as constants throughout the zones.
Visualization tool	Computer software which allows users to turn numbers into pictures. Such a tool helps users identify trends and abnormalities in data sets.
Visualization window	The window where plots are displayed. MeshTV supports a maximum of 16 visualization windows. Also referred to as a Vis window.
Widget	At item in MeshTV’s GUI. Widgets include buttons, text fields, menu items, and scrollable lists, for example.

Zone	An area or volume from which meshes are comprised. Zones are polygons or polyhedra with nodes as vertices. Zones are occupied by one or more materials.
-------------	--